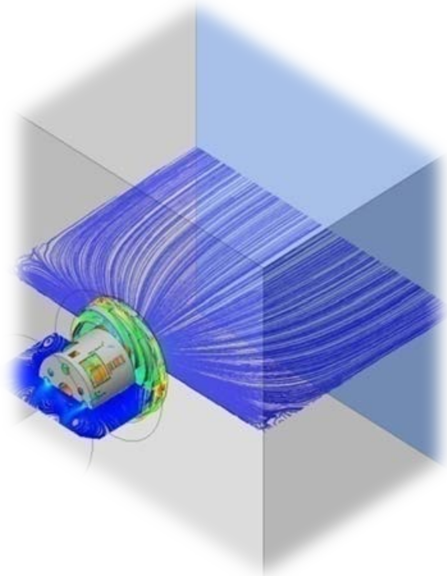
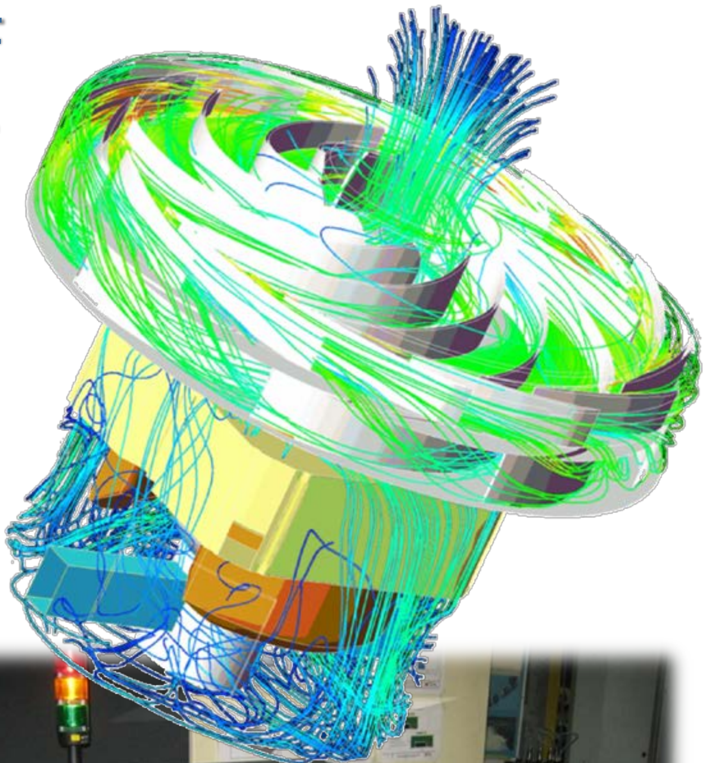
CFD AND EFD IN THE DESIGN PROCESS OF FANS AND BLOWERS

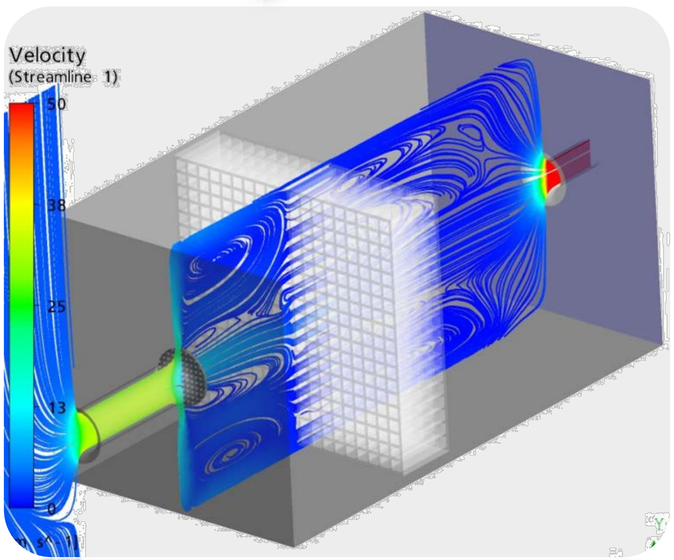
Philipp Epple

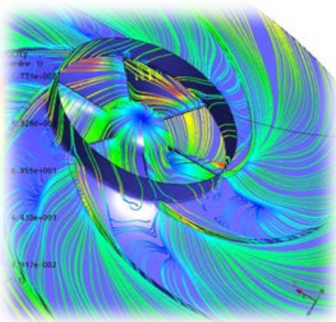
Coburg University
of Applied Sciences

NASA AMS Seminar
September 8th, 2015



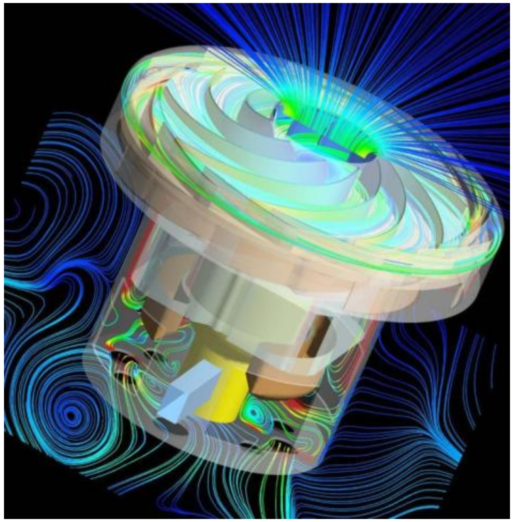






CONTENTS

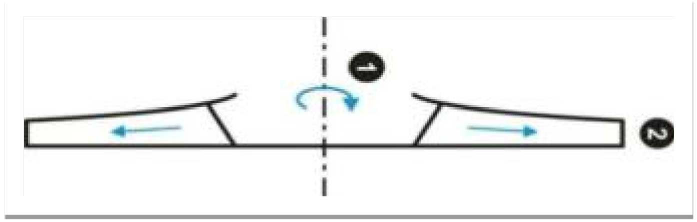
- NEED FOR TFD, CFD AND EFD
- CASE STUDY I:
  SLOTTED DIFFUSER
- CASE STUDY II:
  TORQUE-SPEED CHARACTERISTIC
- CASE STUDY III:
  COMPACT TEST RIG DESIGN

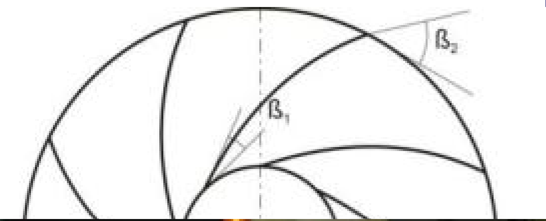


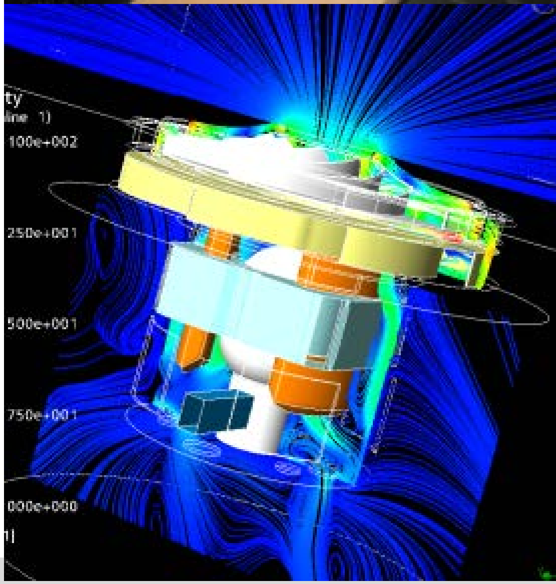
NEED FOR (TFD AND) CFD AND EFD

COBURG UNIVERSITY of applied sciences and arts

$$N = V^P$$





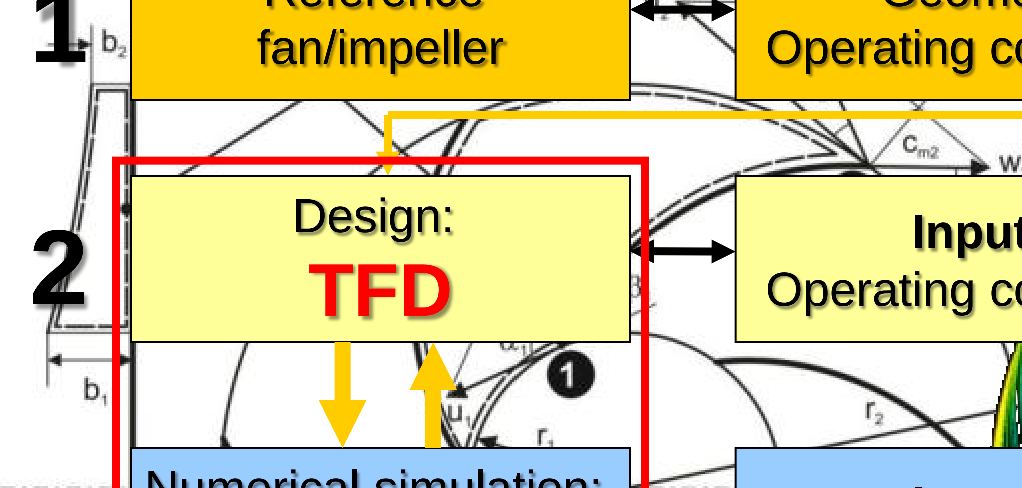


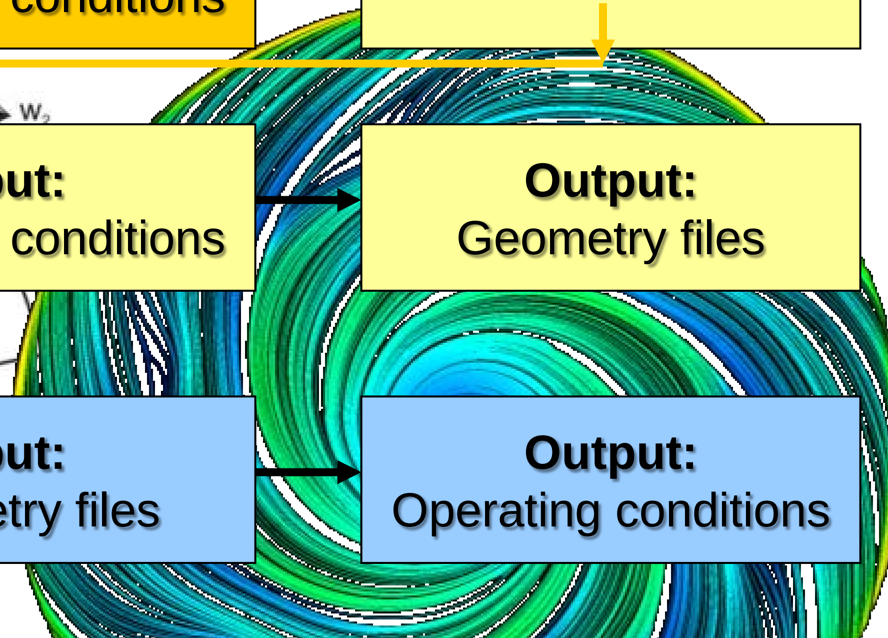
N = number of CFD runs or experiments
P = number of parameters (e.g. diameter, blade number etc.)
V = number of variations or values for each parameter

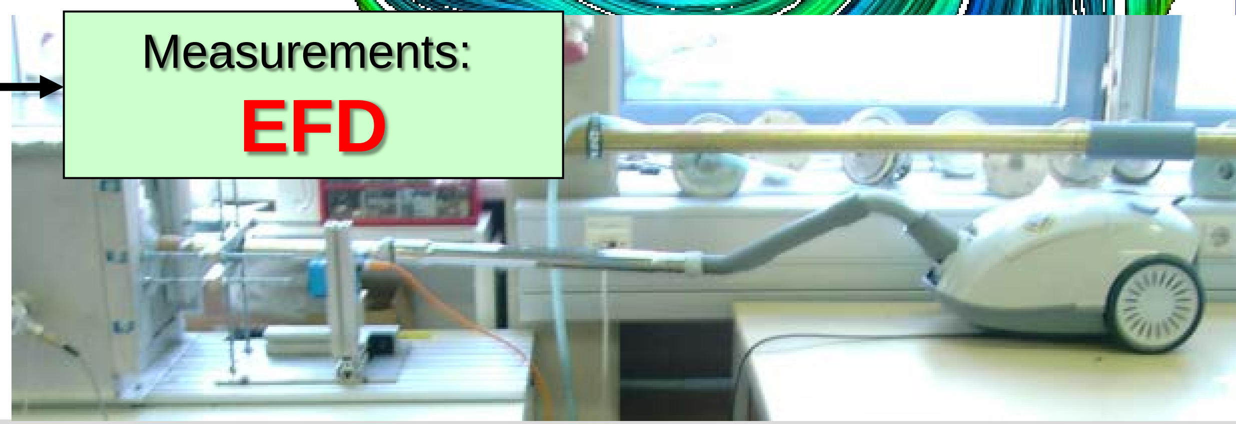
V	**2**	**3**	**4**	**5**
P	**N**	**N**	**N**	**N**
1	2	3	4	5
2	4	9	16	25
3	8	27	64	125
4	16	81	256	625
5	32	243	1.024	3.125
6	64	729	4.096	15.625
7	128	2.187	16.384	78.125
8	256	6.561	65.536	390.625
9	512	19.683	262.144	1.953.125
10	1.024	59.049	1.048.576	9.765.625

(TFD)-CFD-EFD INTEGRATED APPROACH

COBURG UNIVERSITY of applied sciences and arts







1 Reference fan/impeller ↔ Geometry Operating conditions → Design goals

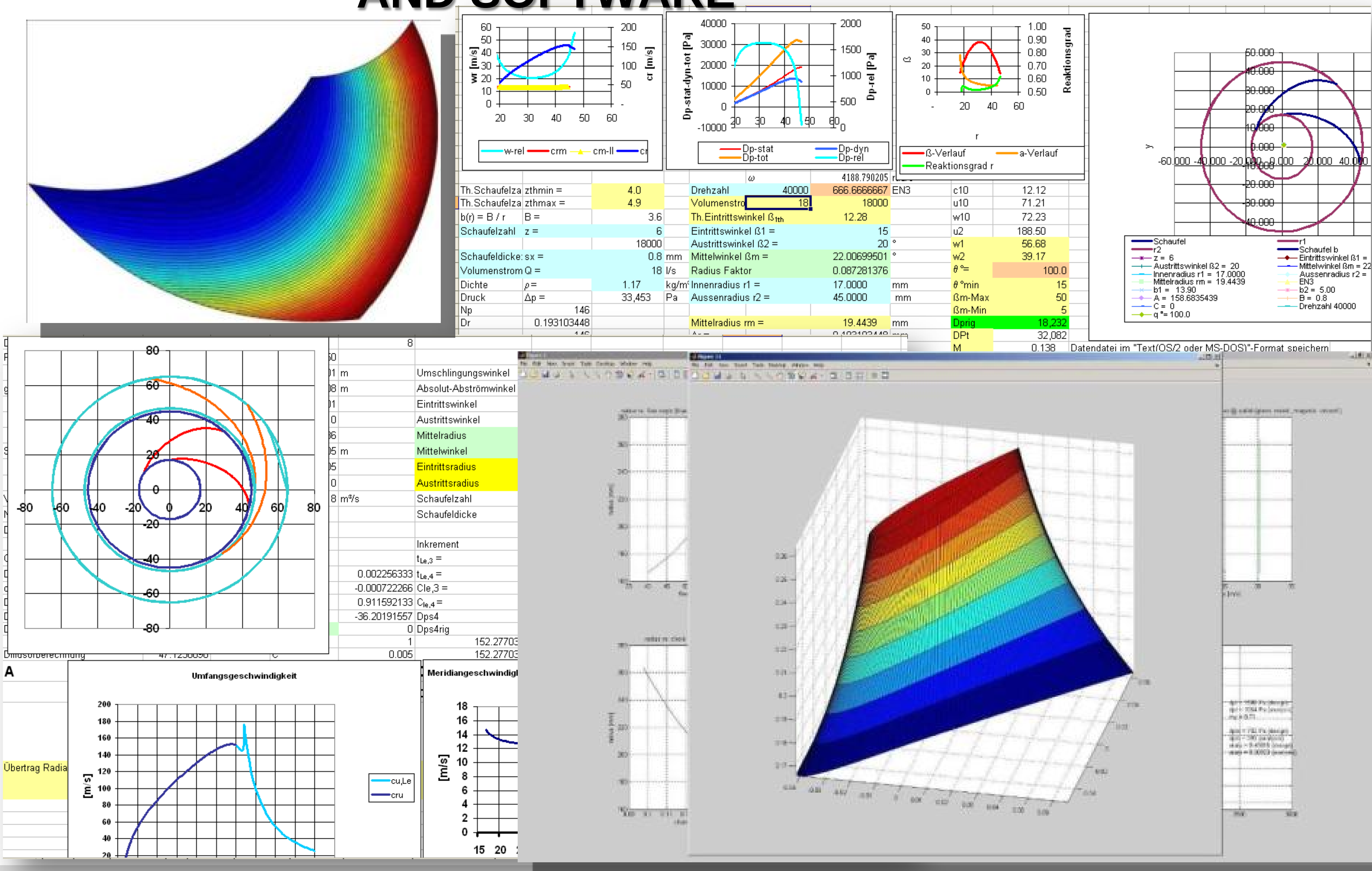
2 Design: **TFD** ↔ **Input:** Operating conditions → **Output:** Geometry files

3 Numerical simulation: **CFD** ↔ **Input:** Geometry files → **Output:** Operating conditions

4 Rapid Prototyping ↔ Measurements: **EFD**

5 Final product

TFD: DESIGN PROCESS AND SOFTWARE



CFD: EMMY HPC CLUSTER RRZE ERLANGEN

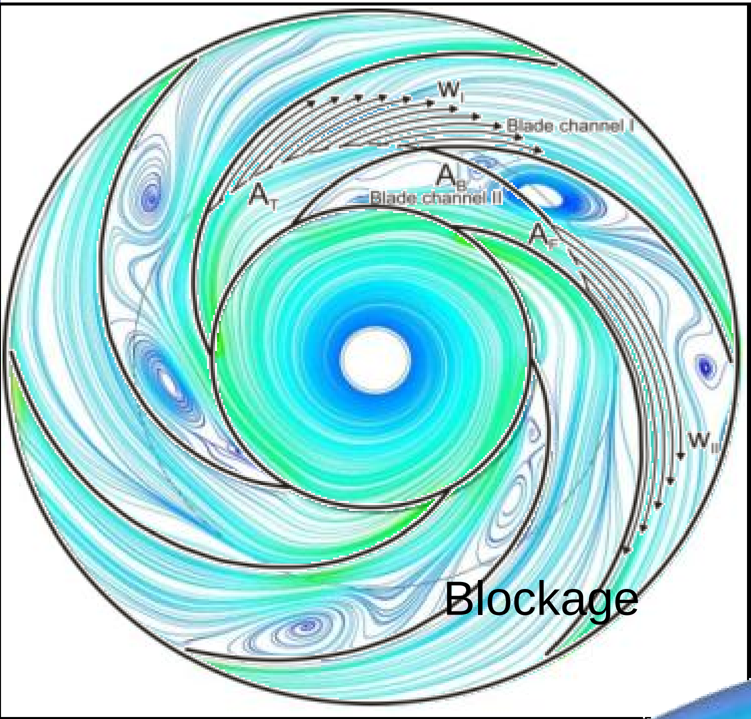
COBURG UNIVERSITY
of applied sciences and arts

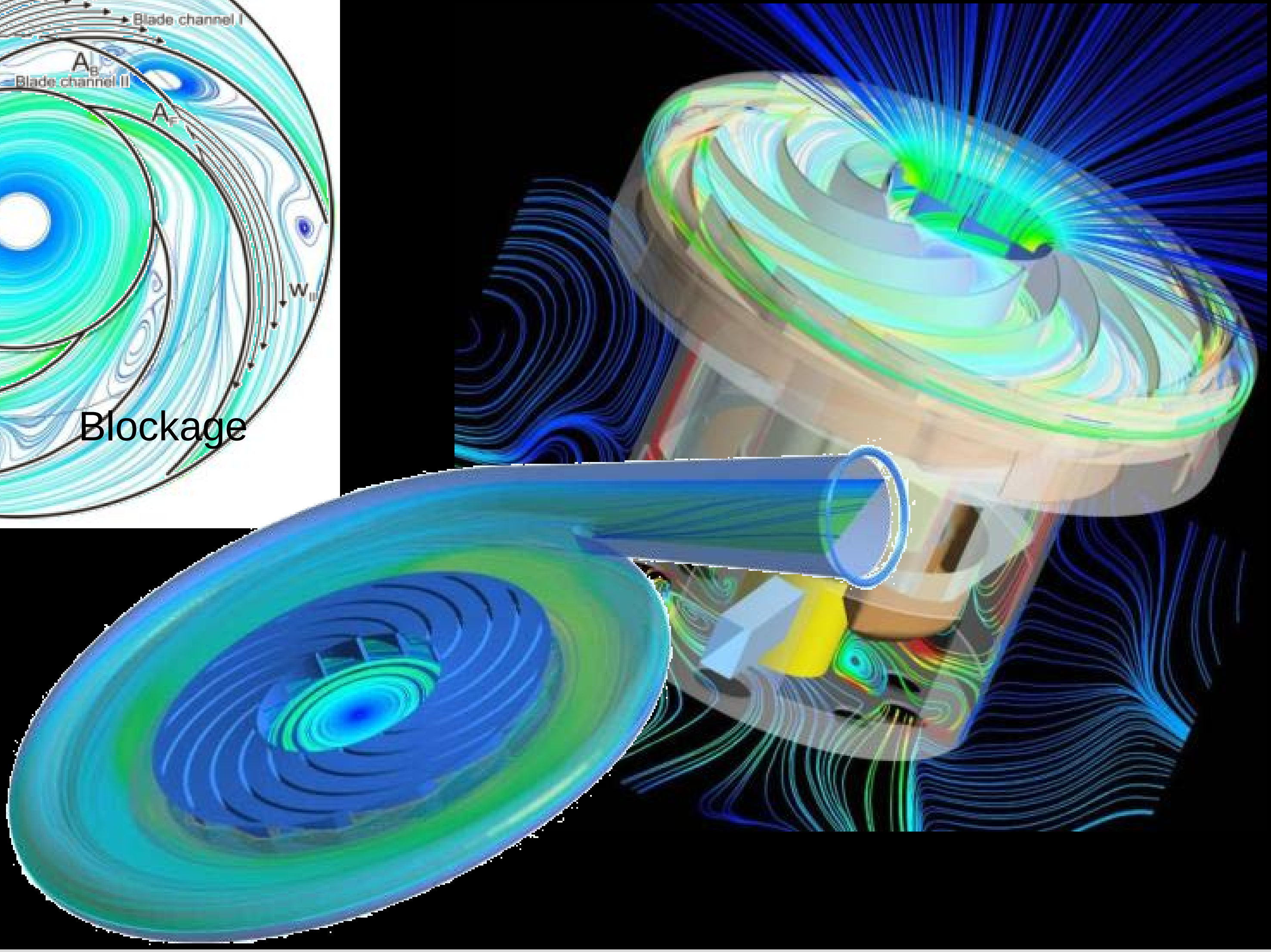
Emmy Cluster

The RRZE's Emmy cluster (NEC) is a high-performance compute resource with high speed interconnect. It is intended for distributed-memory (MPI) or hybrid parallel programs with medium to high communication requirements.

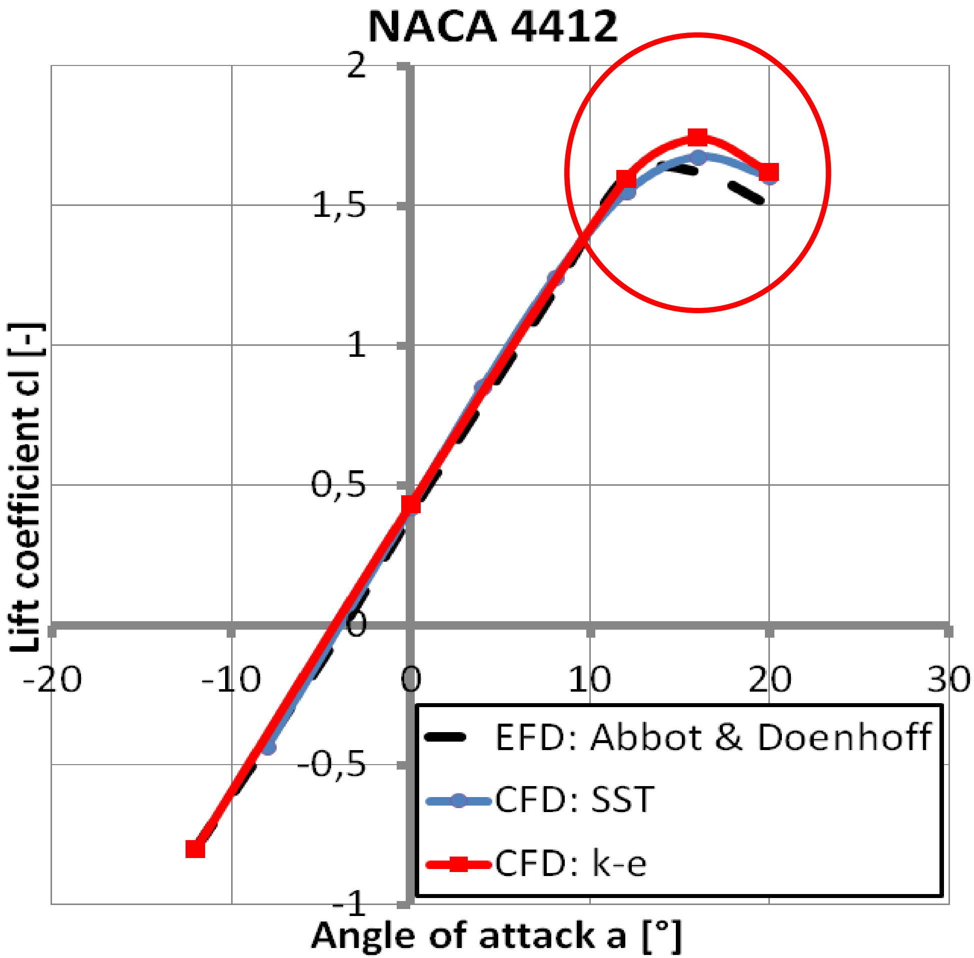
- 560 compute nodes, each with two Xeon 2660v2 "Ivy Bridge" chips (10 cores per chip + SMT) running at 2.2 GHz with 25 MB Shared Cache per chip and 64 GB of RAM
- 2 frontend nodes with the same CPUs as the nodes.
- 16 Intel Xeon Phi coprocessors and 16 Nvidia K20 GPGPUs spread over 16 compute nodes
- parallel filesystem (LXFS) with capacity of 400 TB and an aggregated parallel I/O bandwidth of > 7000 MB/s
- Infiniband interconnect fabric with 40 GBit/s bandwith per link and direction
- Overall peak performance of ca. 234 TFlop/s (191 TFlop/s LINPACK, using only the CPUs).
- November 2013 on rank 210 of the Top500 list of the most powerful supercomputers worldwide

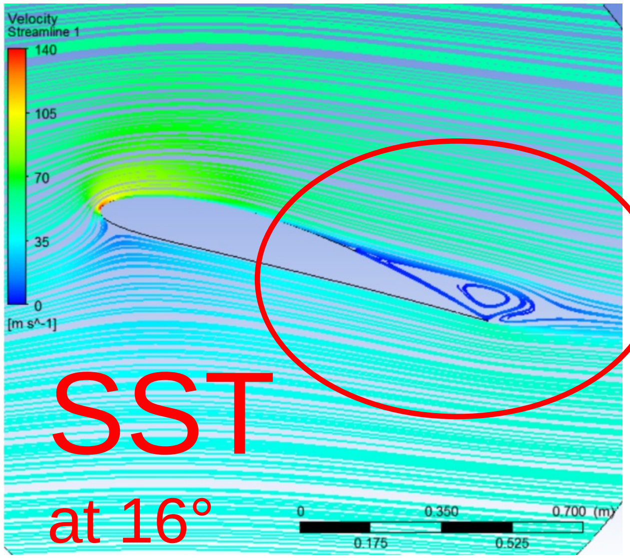
COBURG UNIVERSITY
of applied sciences and arts

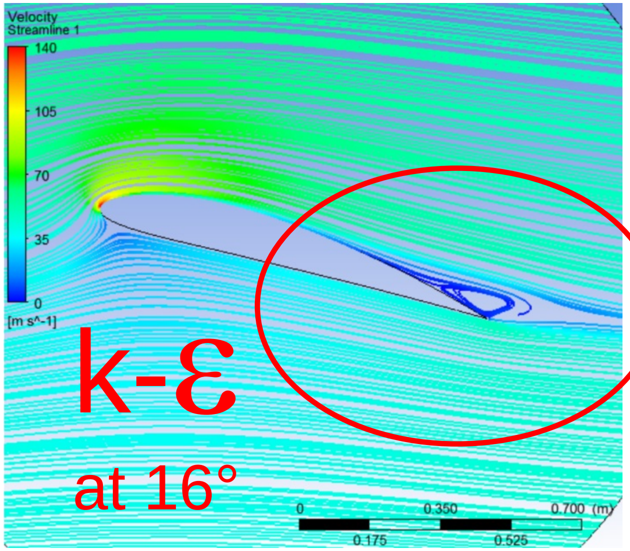




CFD: TURBULENCE MODEL

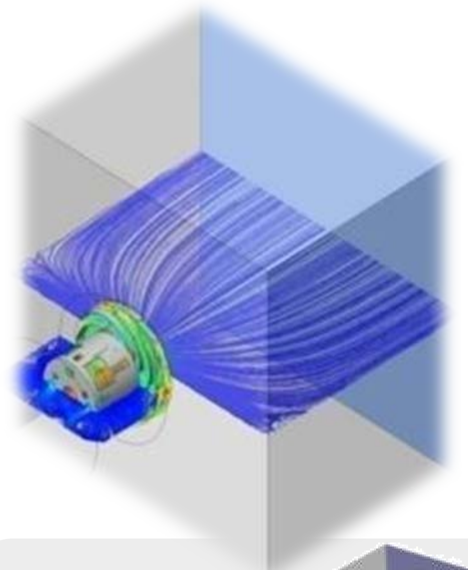


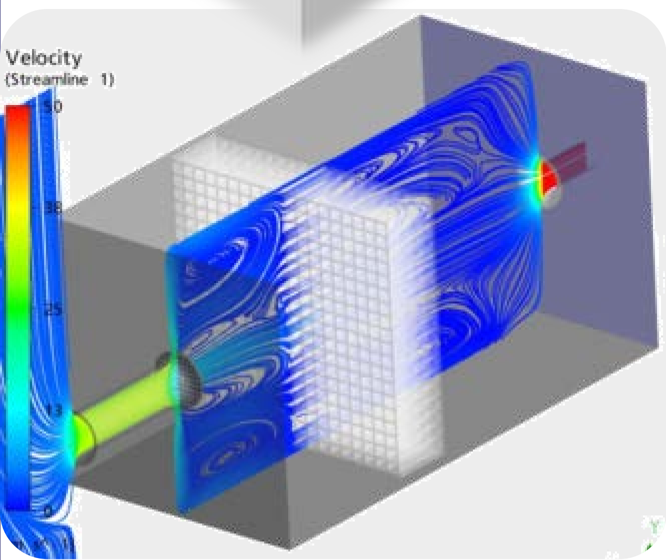


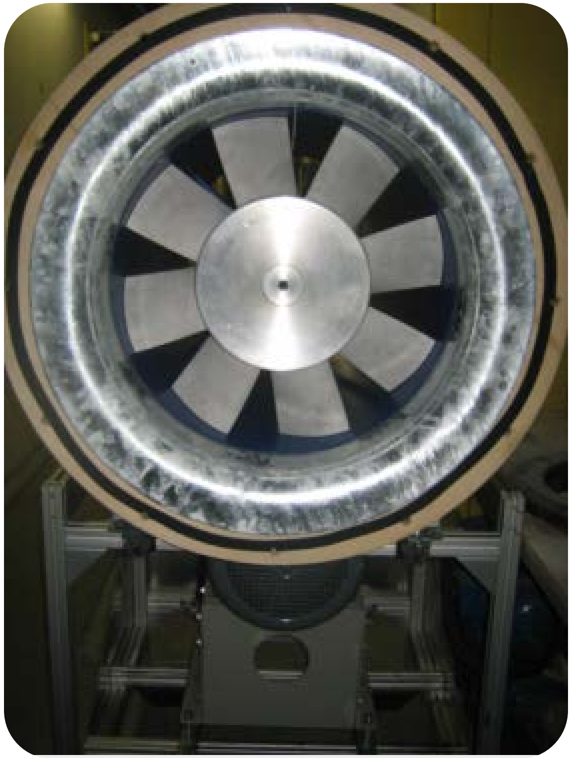


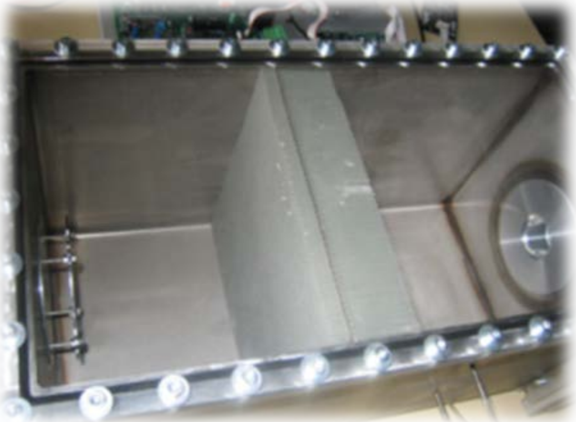
Different results especially near flow detachment or stall.

EFD: TEST RIGS

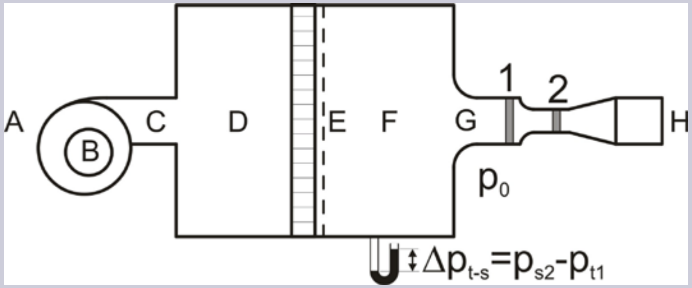


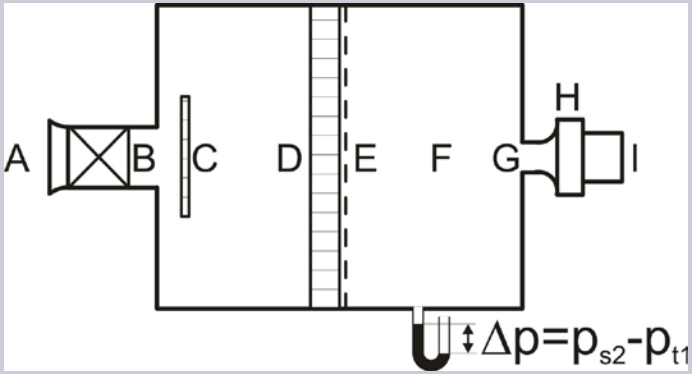






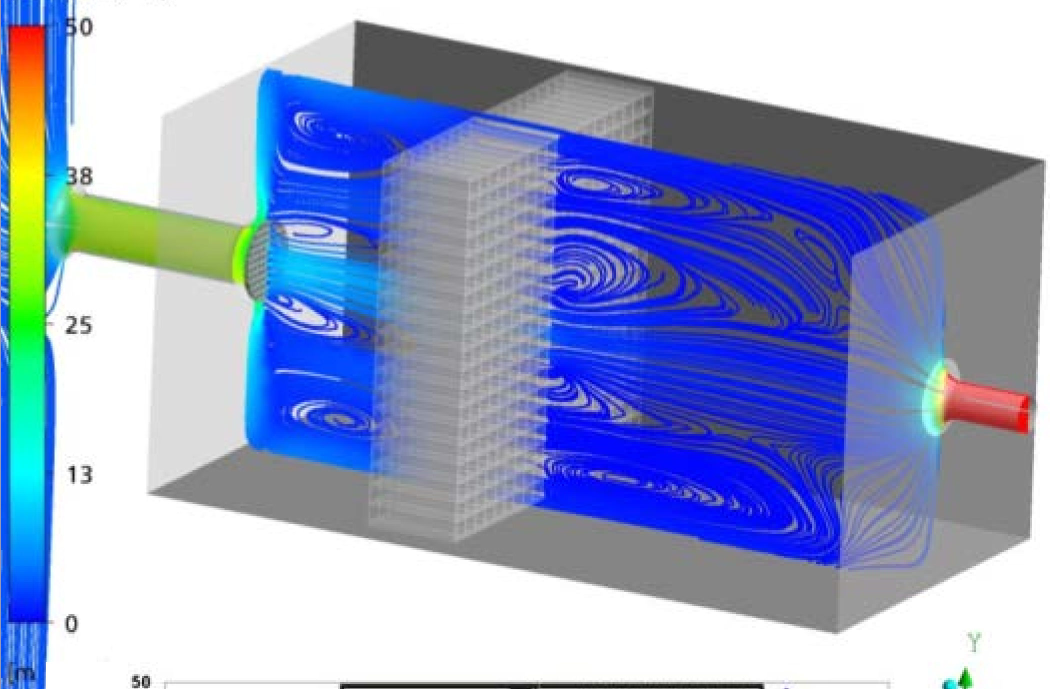
PRESSURE SIDE AND SUCTION SIDE TEST RIGS

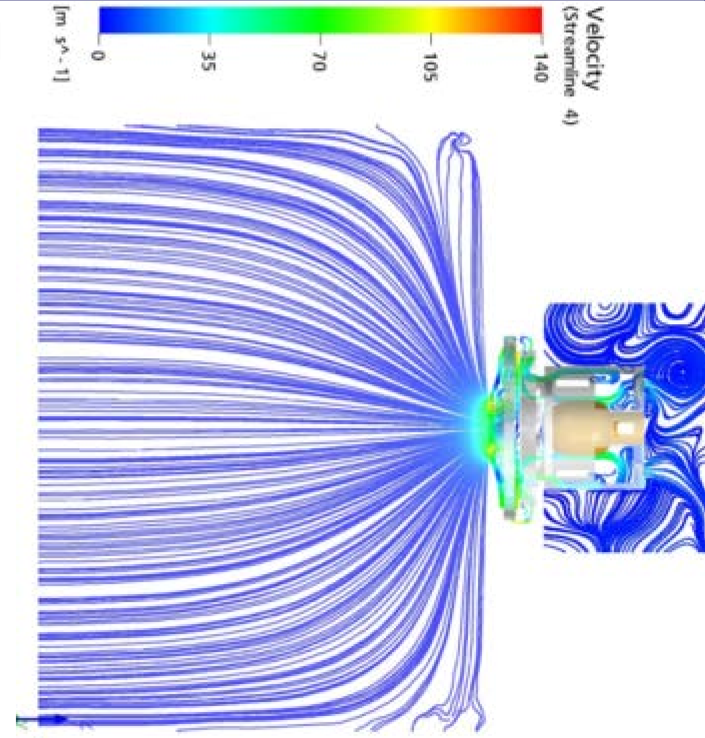
<table>
<tr><th></th><th>Pressure side</th><th>Suction side</th></tr>
<tr><td>Fan and Blower Test Rig</td><td>

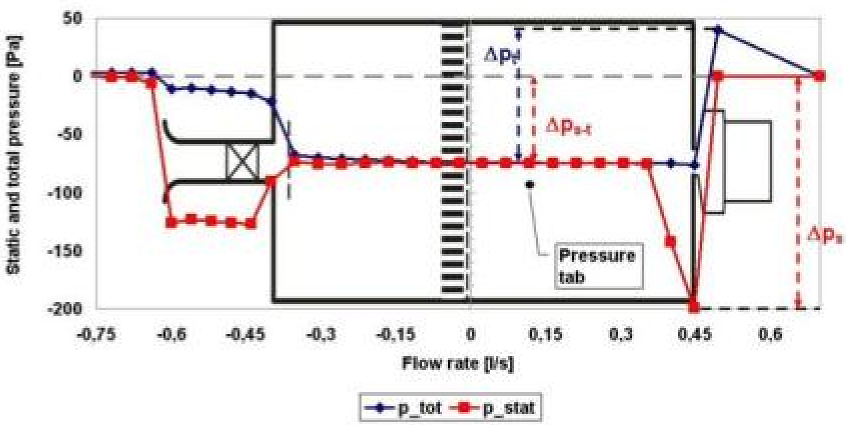
</td><td>

</td></tr>
</table>

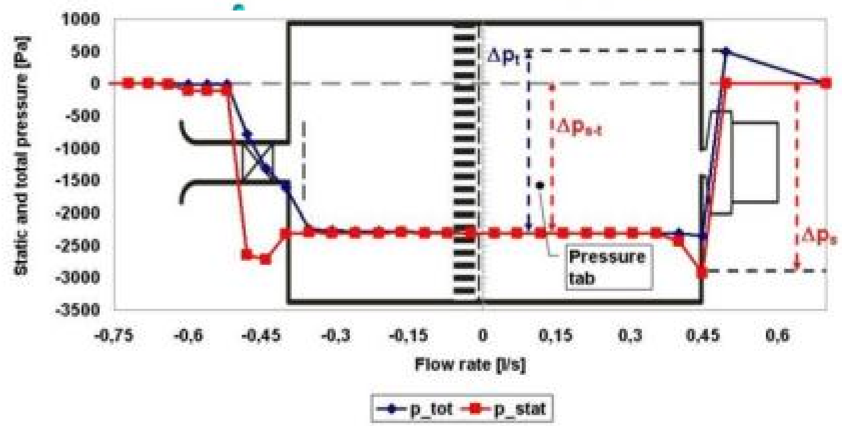
CFD IN TEST RIG DESIGN







Open throttle

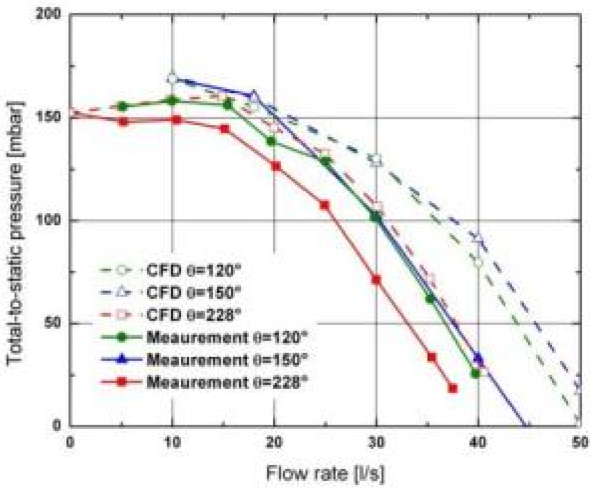


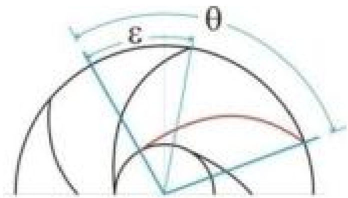
Partially closed throttle

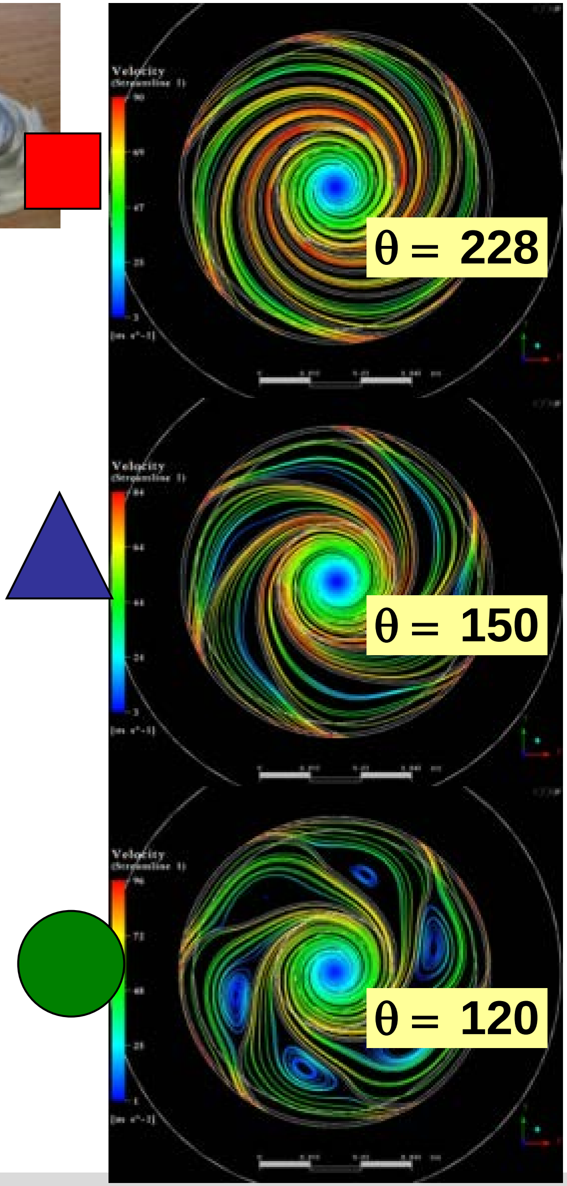
CFD helping to build better test rigs

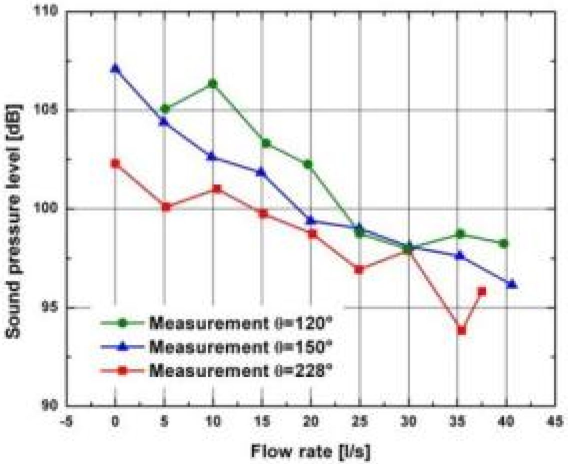
CFD – EFD INCL. SOUND LEVEL

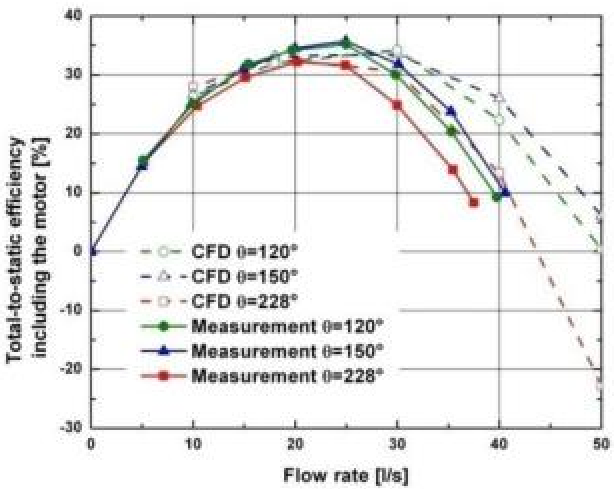
COBURG UNIVERSITY
of applied sciences and arts









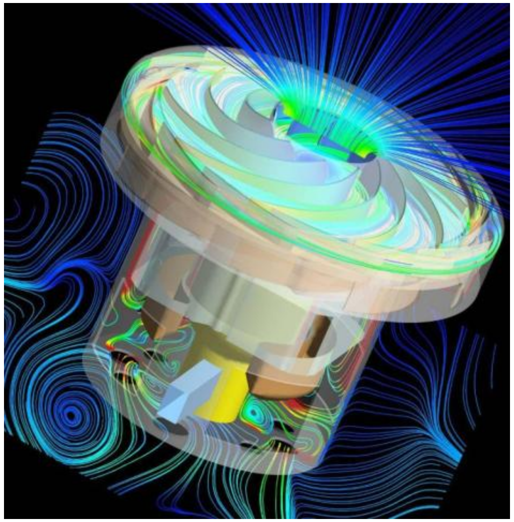


Inlet diameter	d_1	34	[mm]
Outer diameter	d_2	90	[mm]
Inlet height	b_1	14	[mm]
Outlet heigth	b_2	5	[mm]
Inlet angle	$ß_1$	15	[°]
Outlet angle	$ß_2$	20	[°]
Blade thickness	s	0,8	[mm]
Blades	z	6	

Table : Impeller main dimensions

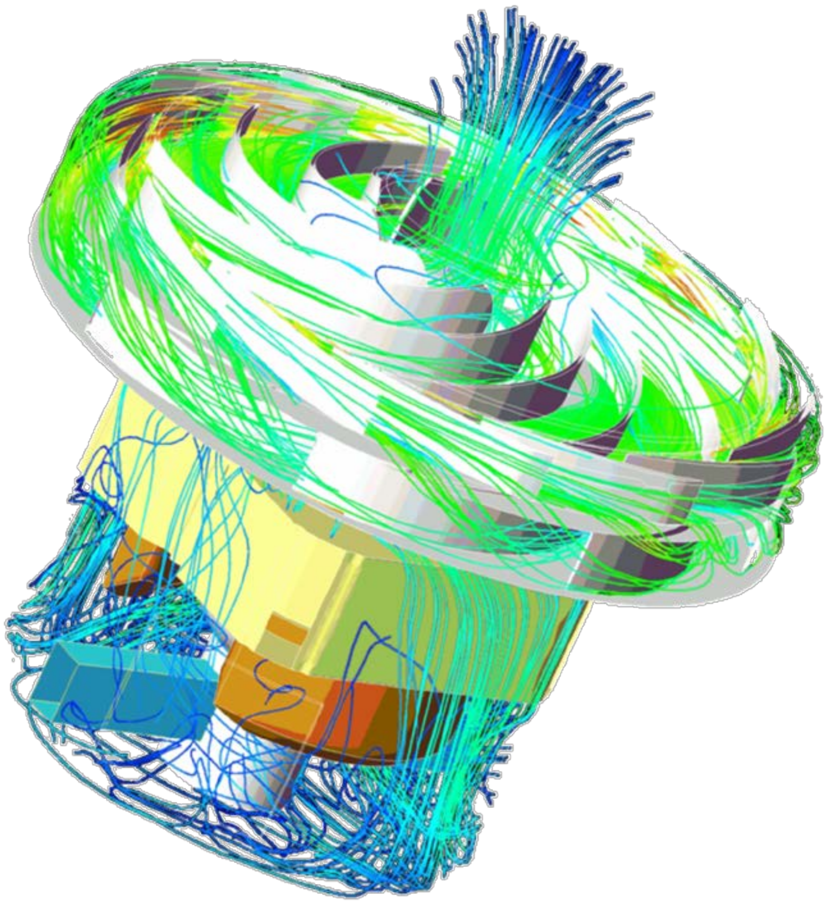
CONTENTS

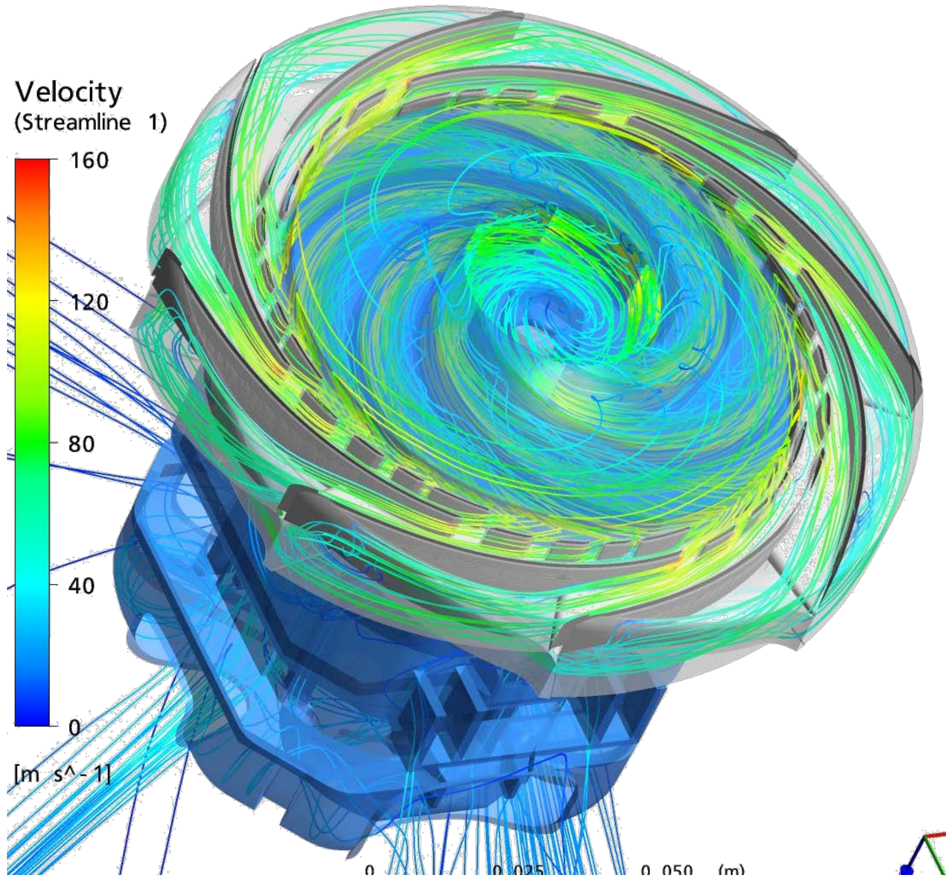
- NEED FOR TFD, CFD AND EFD
- CASE STUDY I:
  SLOTTED DIFFUSER
- CASE STUDY II:
  TORQUE-SPEED CHARACTERISTIC
- CASE STUDY III:
  COMPACT TEST RIG DESIGN



Case study I

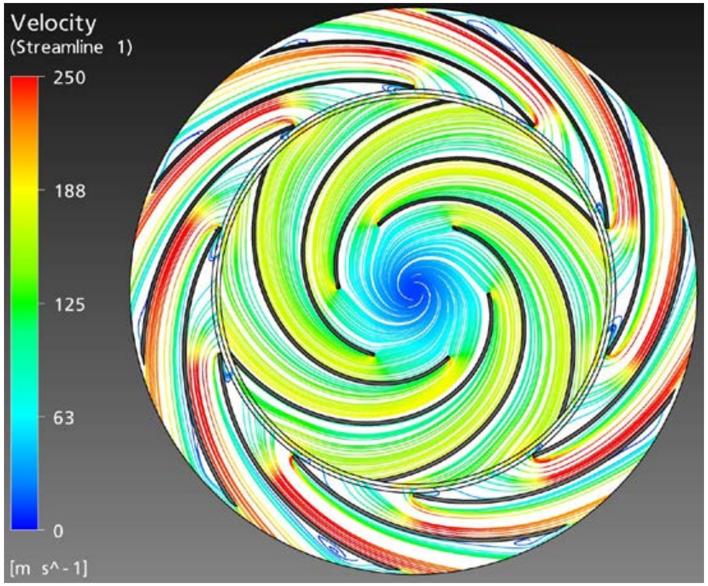
THE INFLUENCE OF SLOTTED GUIDE VANES ON THE PERFORMANCE OF RADIAL DIFFUSERS

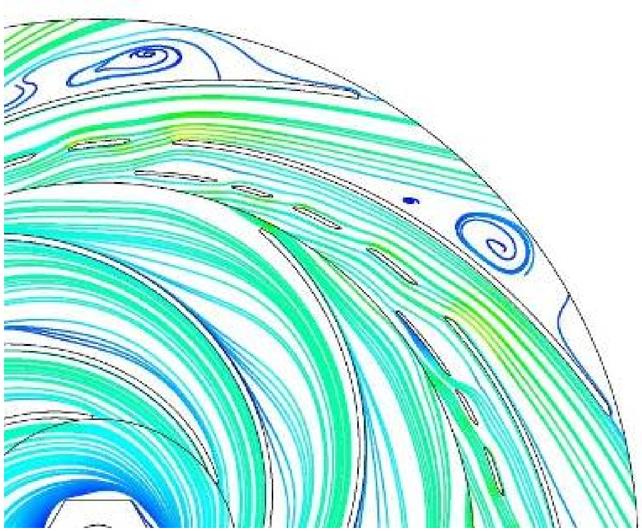


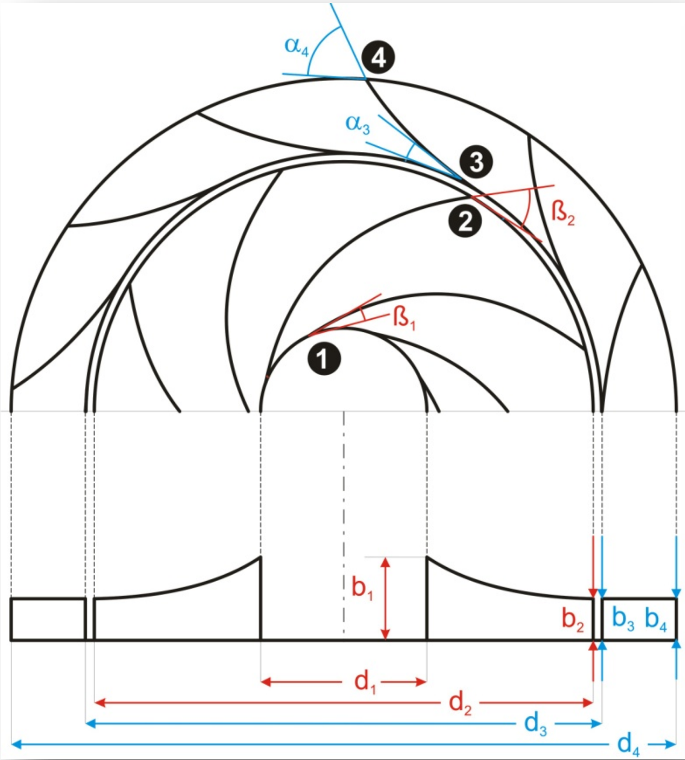


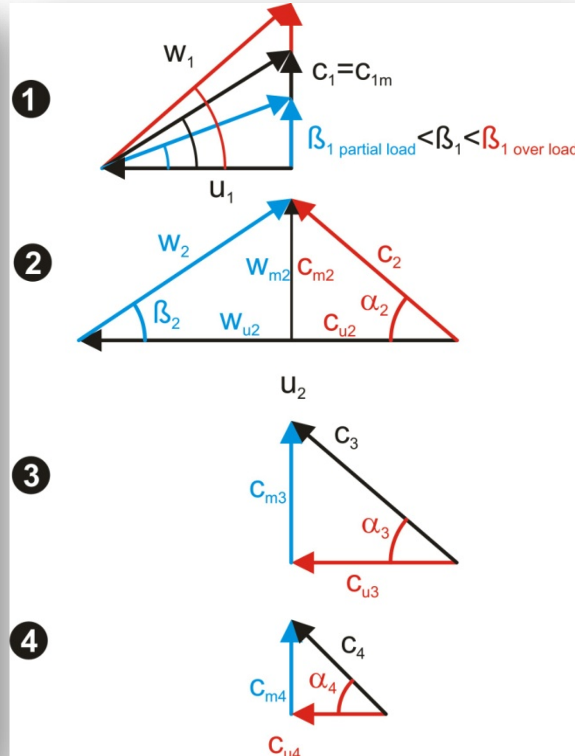
Contents

1. Diffuser working principles
2. Impeller flow regimes
3. Diffuser flow regimes and choking - examples
4. Working principle of slotted vanes
5. CFD verification
6. Measurements
7. Results and conclusions







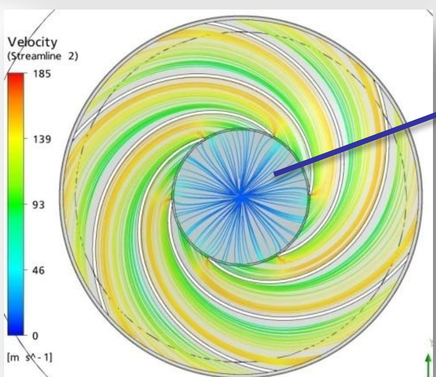


1 Impeller inlet:
- u_1: inlet peripheral velocity
- c_1: inlet absolute velocity
- c_{m1}: inlet meridian absolute velocity
- w_1: inlet relative velocity
- w_{m1}: inlet meridian relative velocity
- w_{u1}: inlet peripheral relative velocity
- $ß_1$: inlet blade angle
- γ_1: inlet flow angle

2 Impeller outlet:
- u_2: outlet peripheral velocity
- c_2: outlet absolute velocity
- c_{m2}: outlet meridian absolute velocity
- c_{u2}: outlet peripheral absolute velocity
- w_2: outlet relative velocity
- w_{m2}: outlet meridian relative velocity
- w_{u2}: outlet peripheral relative velocity
- α_2: outlet absolute flow angle
- $ß_2$: outlet blade angle

3 Diffuser inlet:
- c_3: inlet absolute velocity
- c_{m3}: inlet meridian absolute velocity
- α_3: inlet absolute flow angle

4 Diffuser outlet:
- c_4: inlet absolute velocity
- c_{m4}: inlet meridian absolute velocity
- α_4: inlet absolute flow angle



Radial entry

$$\vec{c} = \vec{w} + \vec{u}$$

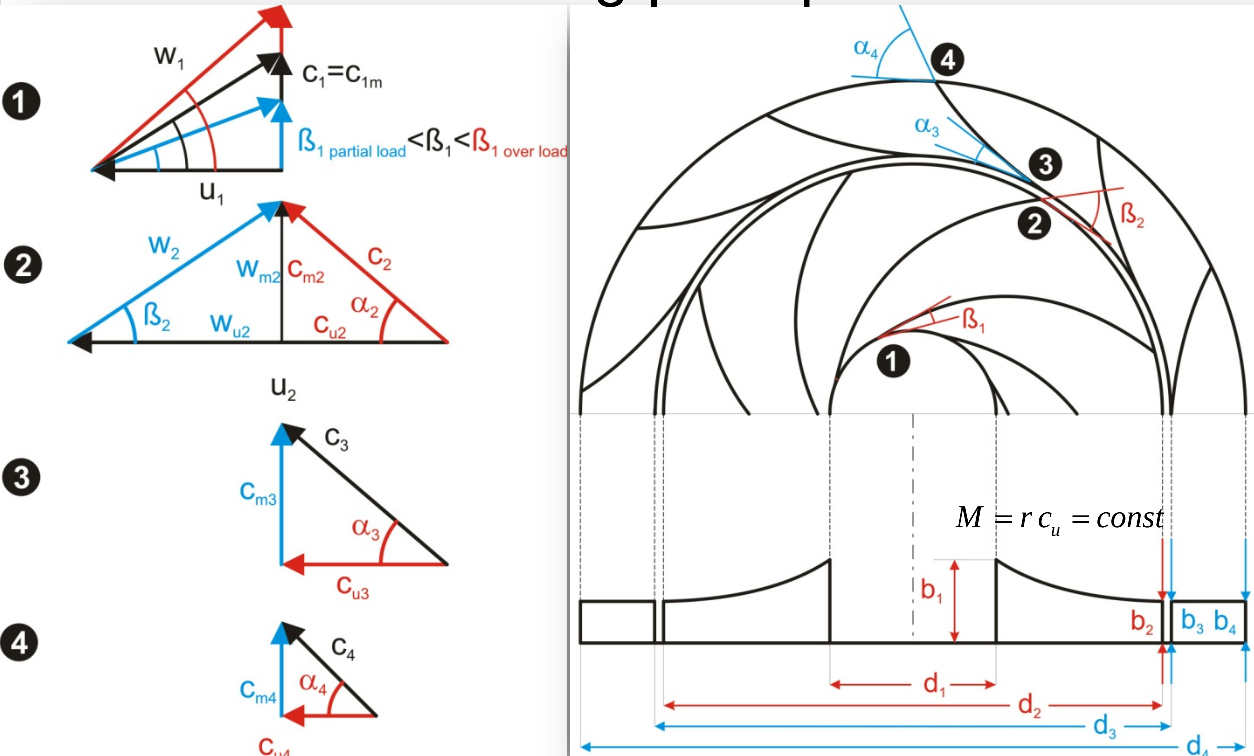
absolute velocity

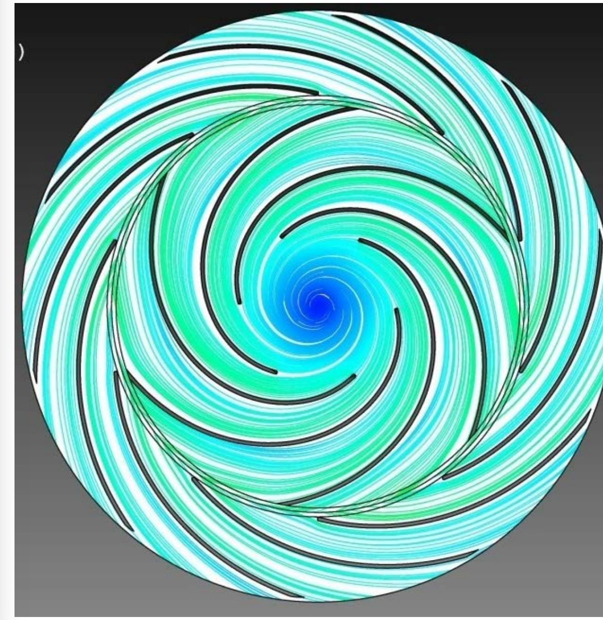
relative velocity

peripheral velocity

Velocity triangles

Working principles

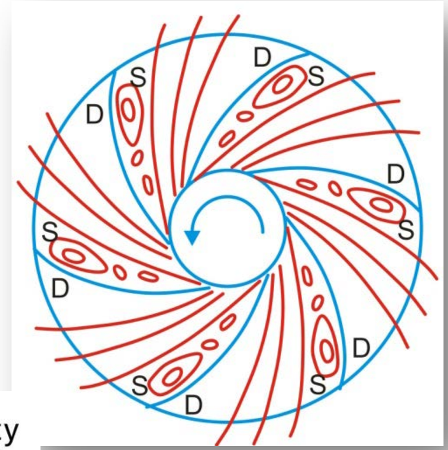


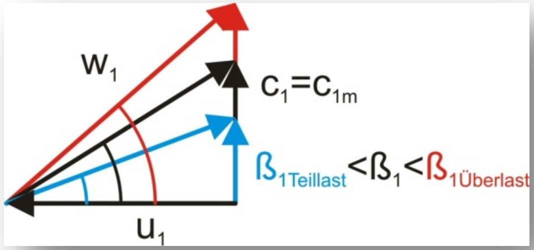


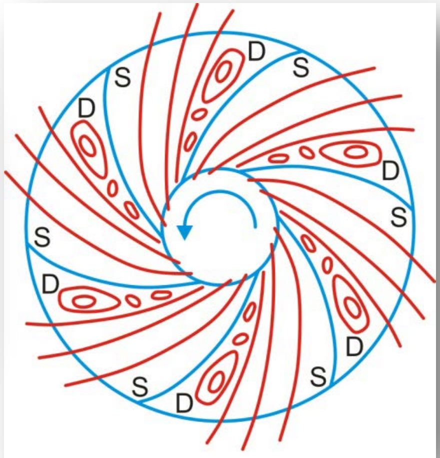
In a diffuser, the pressure increase occurs for two reasons:

1) All diffusers: due to an area increase and hence due to a reduction in the meridian velocity c_m
2) Vaneless diffuser: due to reduction of the tangential velocity component c_u due to the radius increase due to constant angular momentum $M = r\,c_u = const$
3) Vaned diffusers: Different as with the vaneless diffuser, in the vaned diffuser one can reduce the tangential velocity component c_u up to zero, i.e. one can recover all the swirl into static pressure.

Flow Regimes for the Impeller only



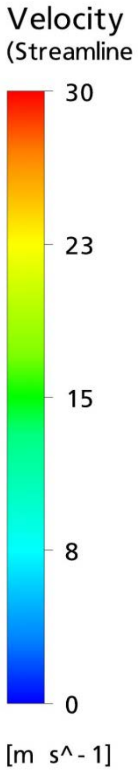


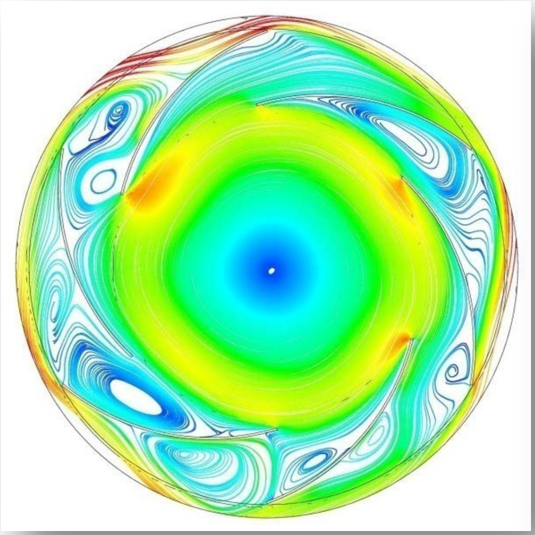


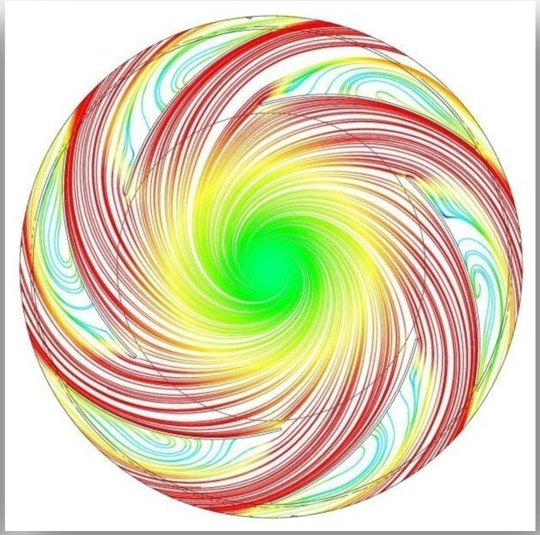
Low flow Q<Qo

Nominal flow Qo

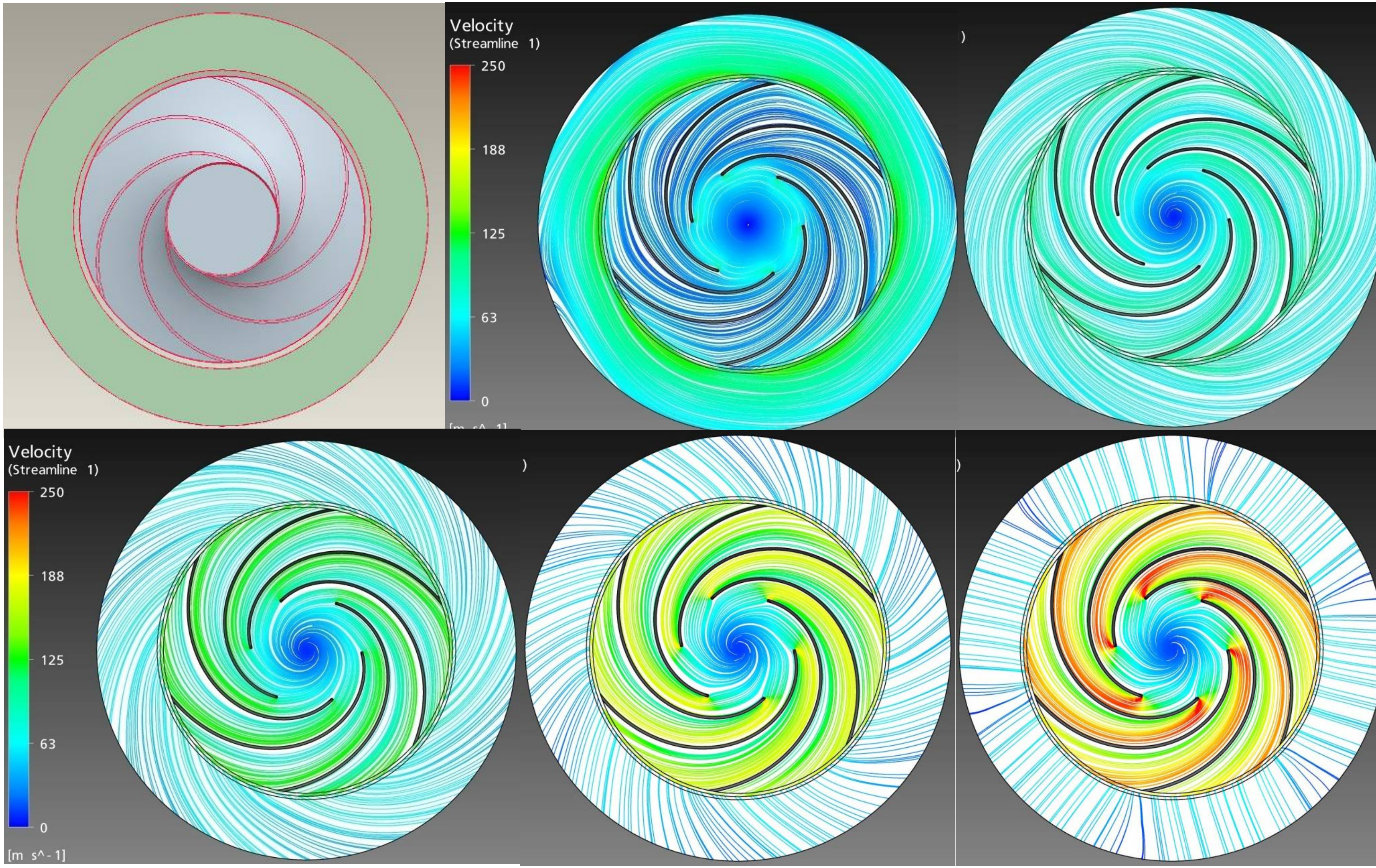
High flow Q>Qo



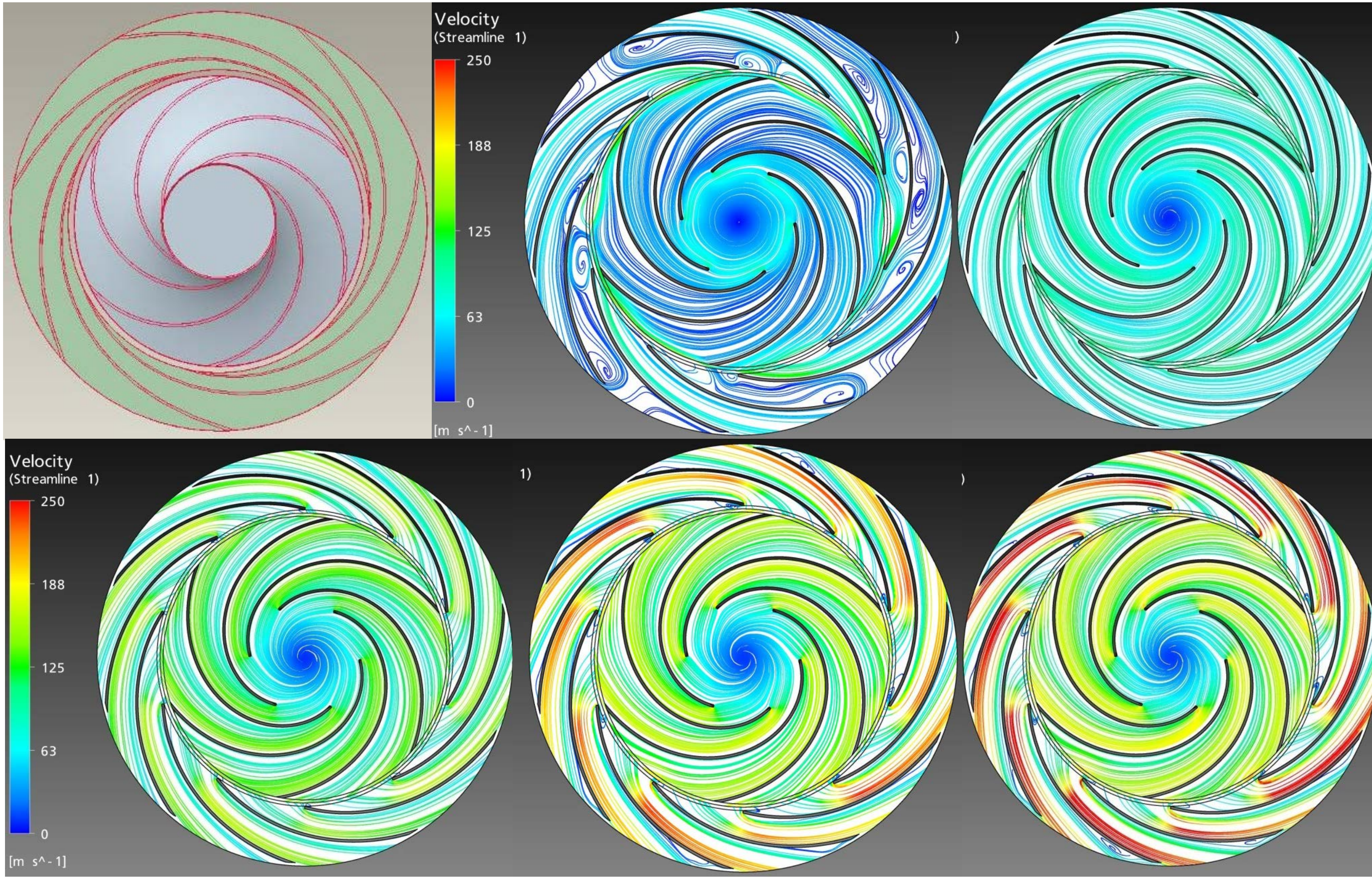




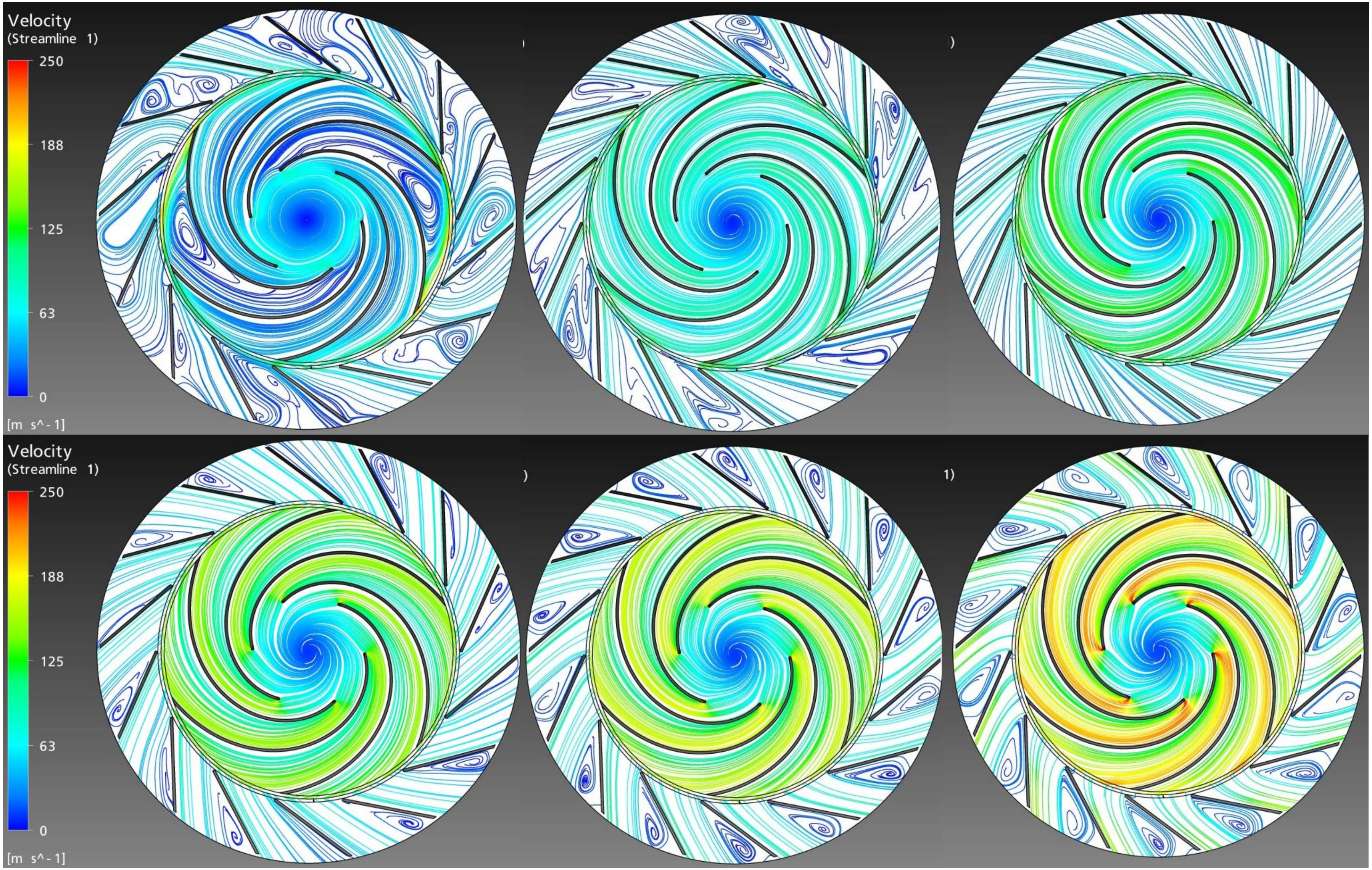
Vaneless Diffuser



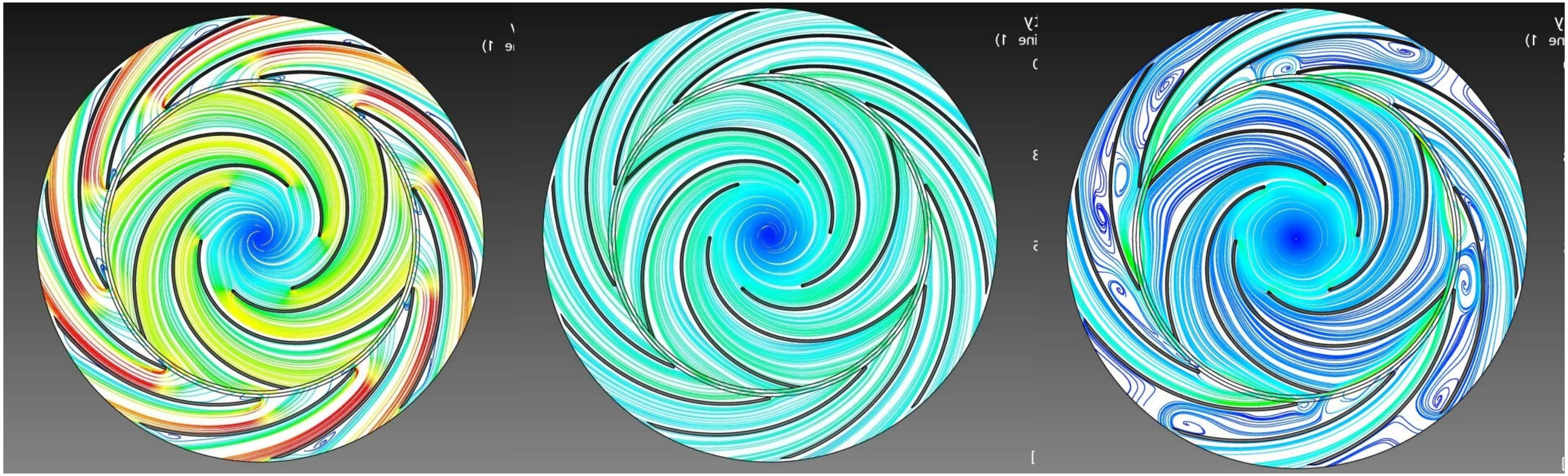
Vaned logarithmic Diffuser

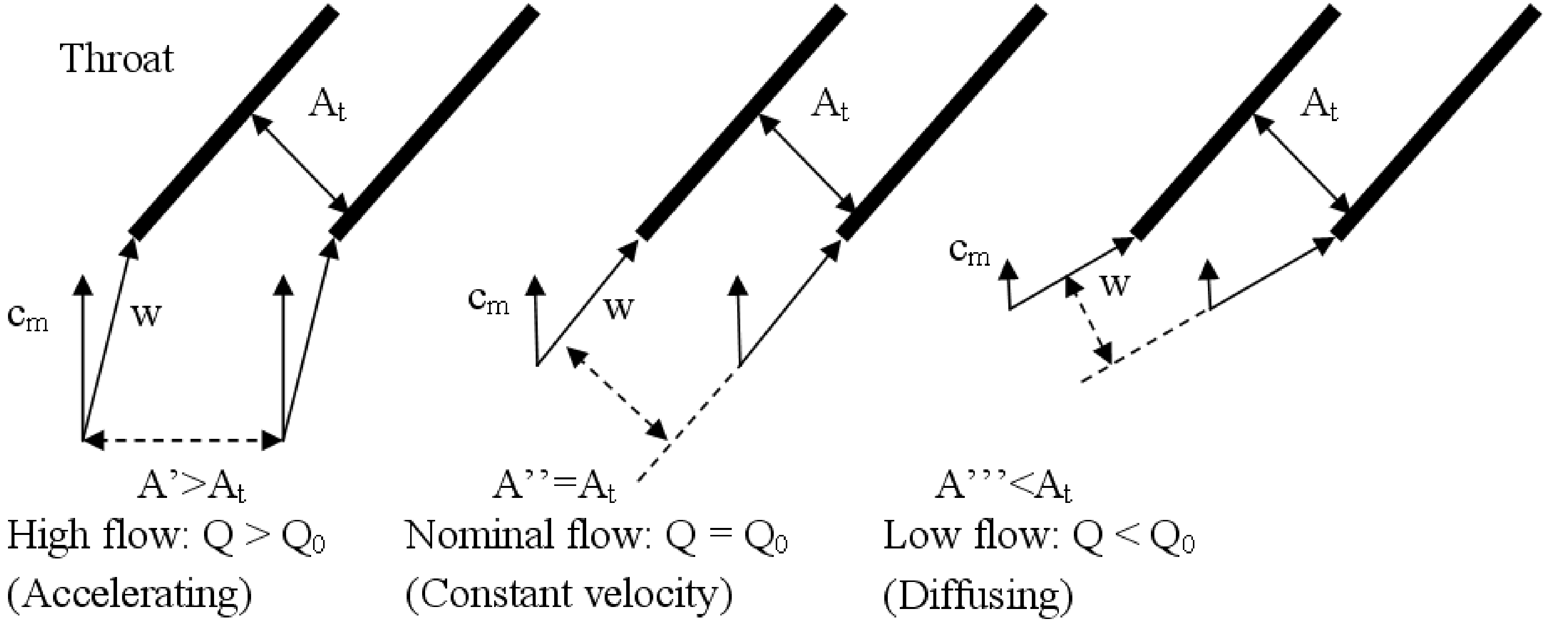


Vaned straight Diffuser

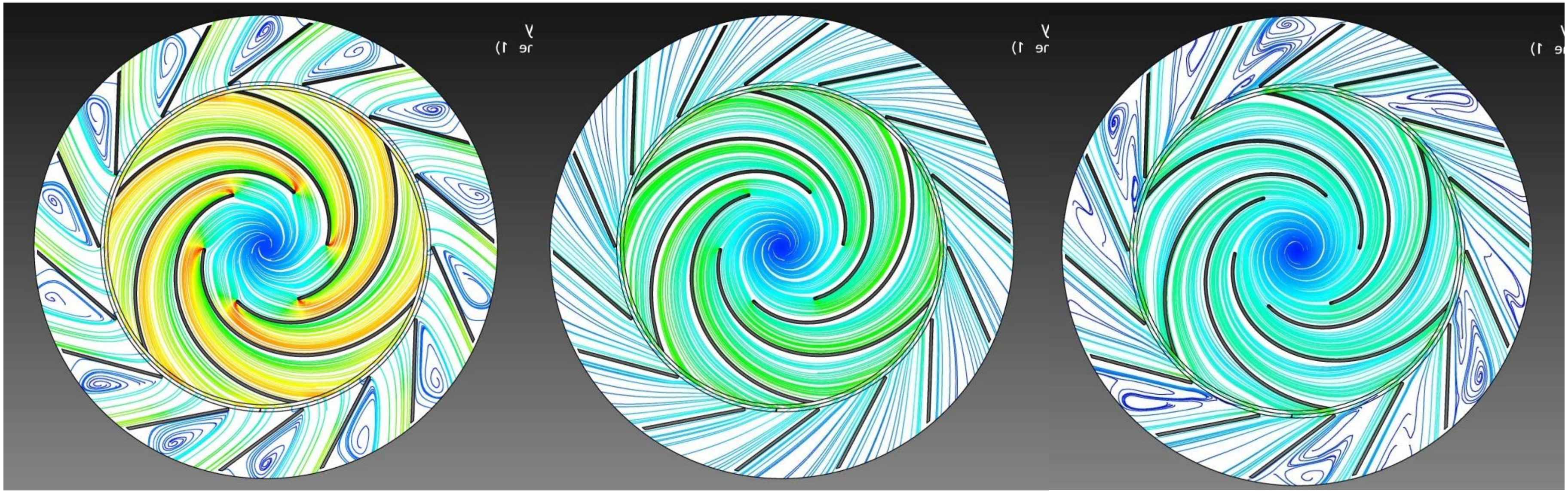


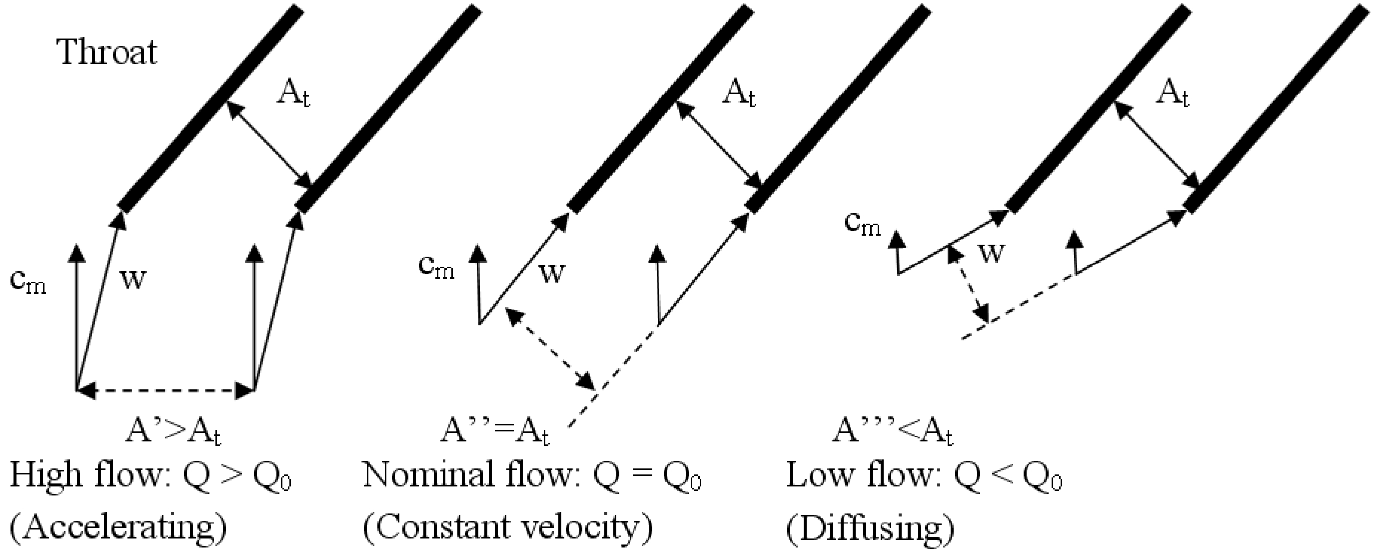
The choked diffuser



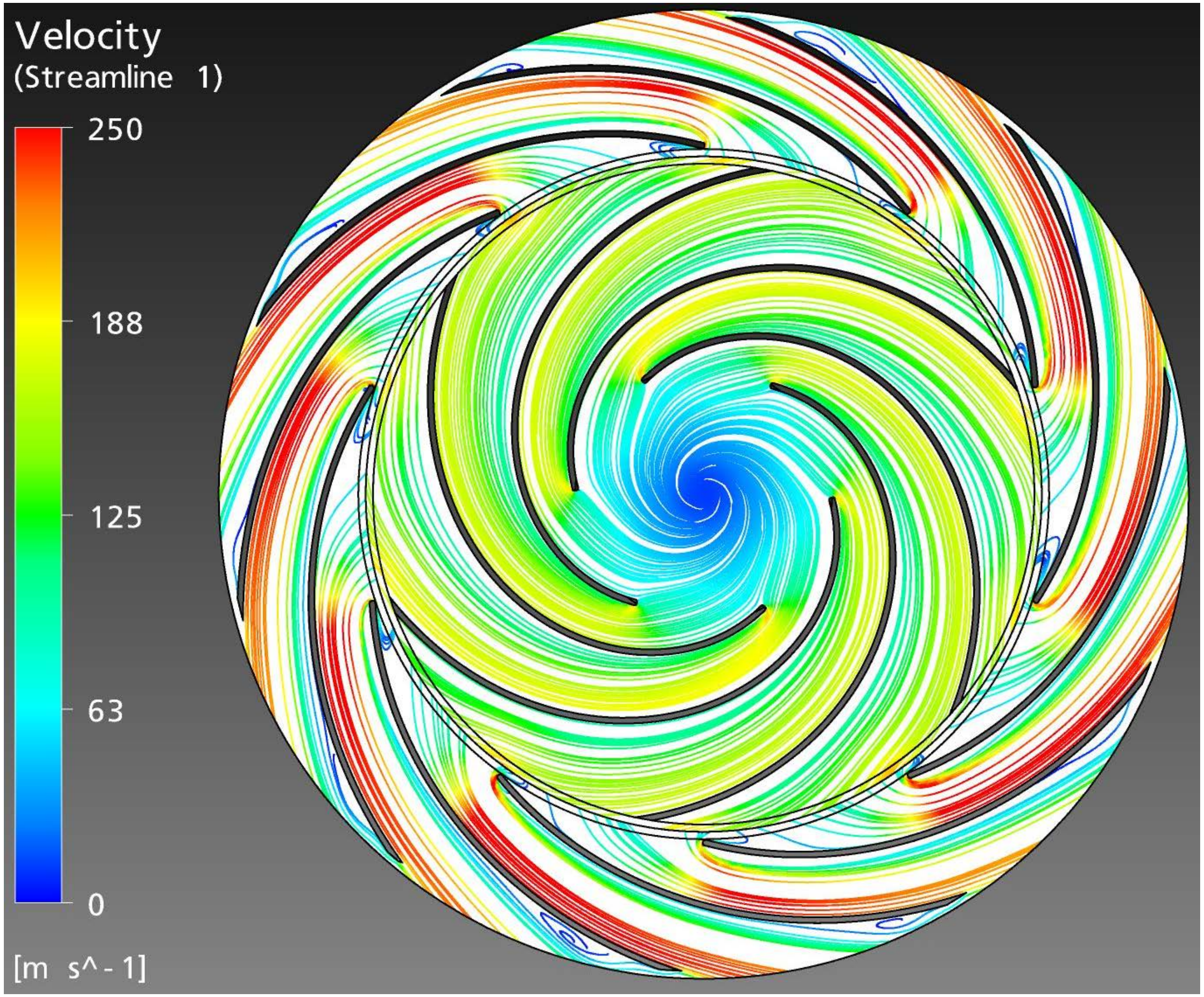


The choked diffuser



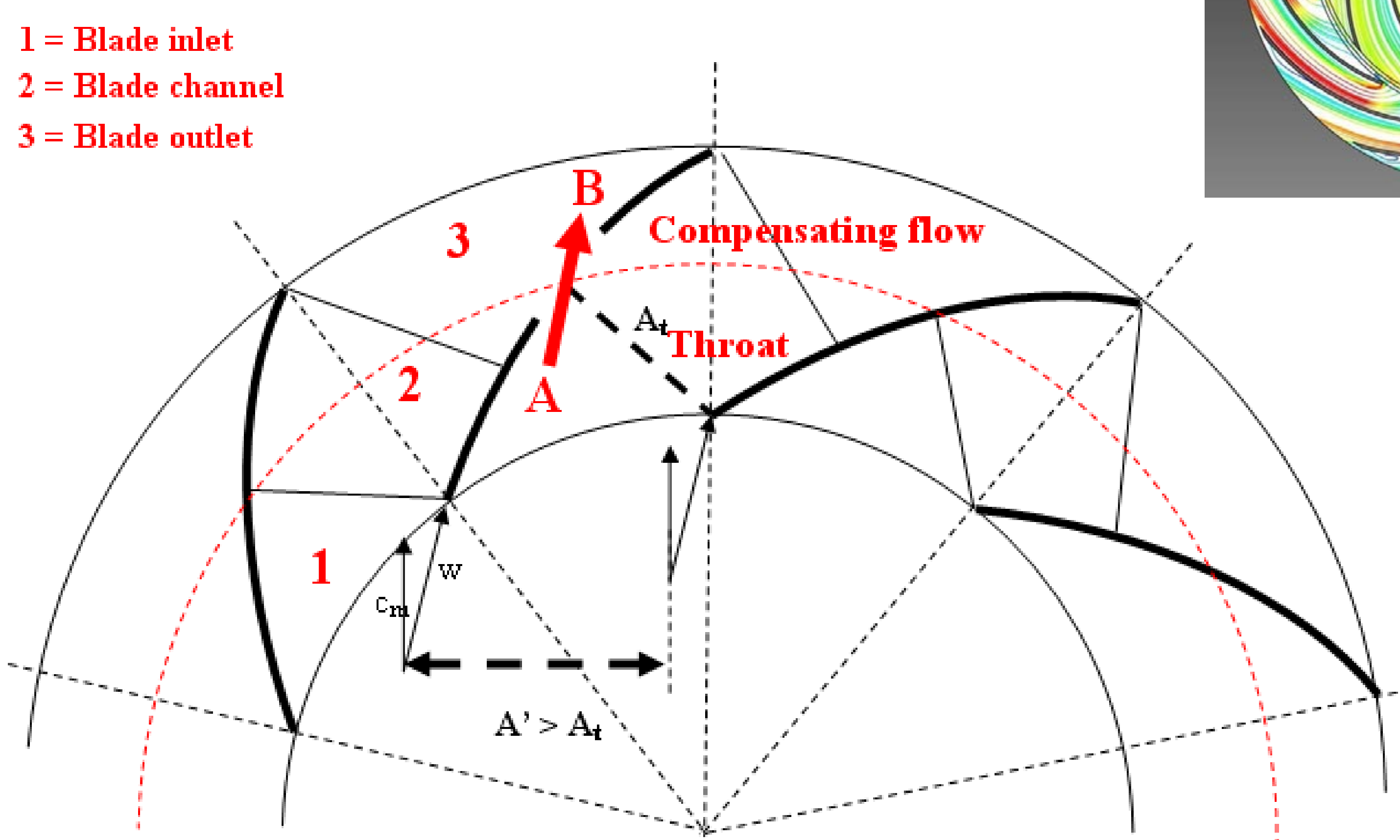


Solution for the choked diffuser?

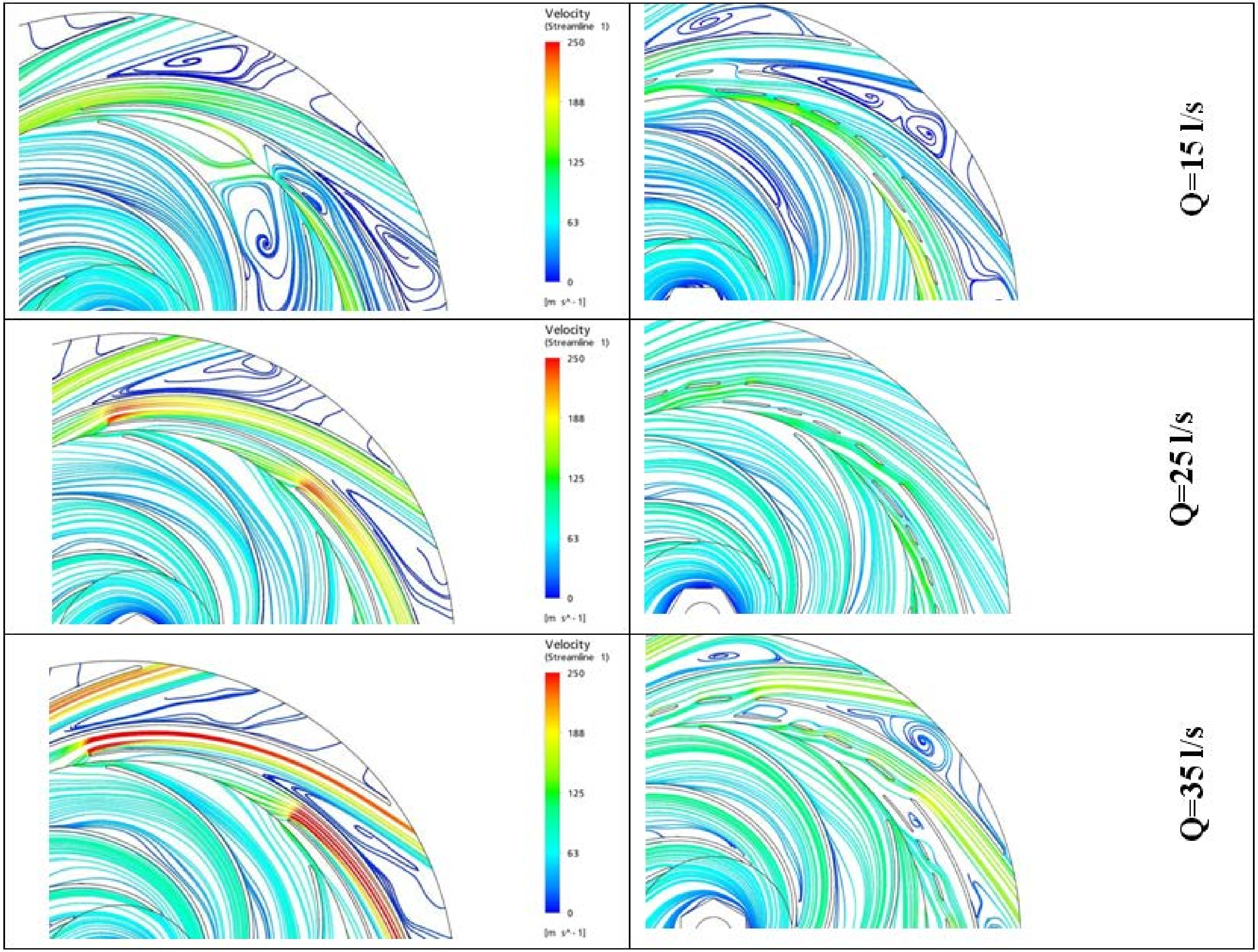


The principle of the slotted diffuser

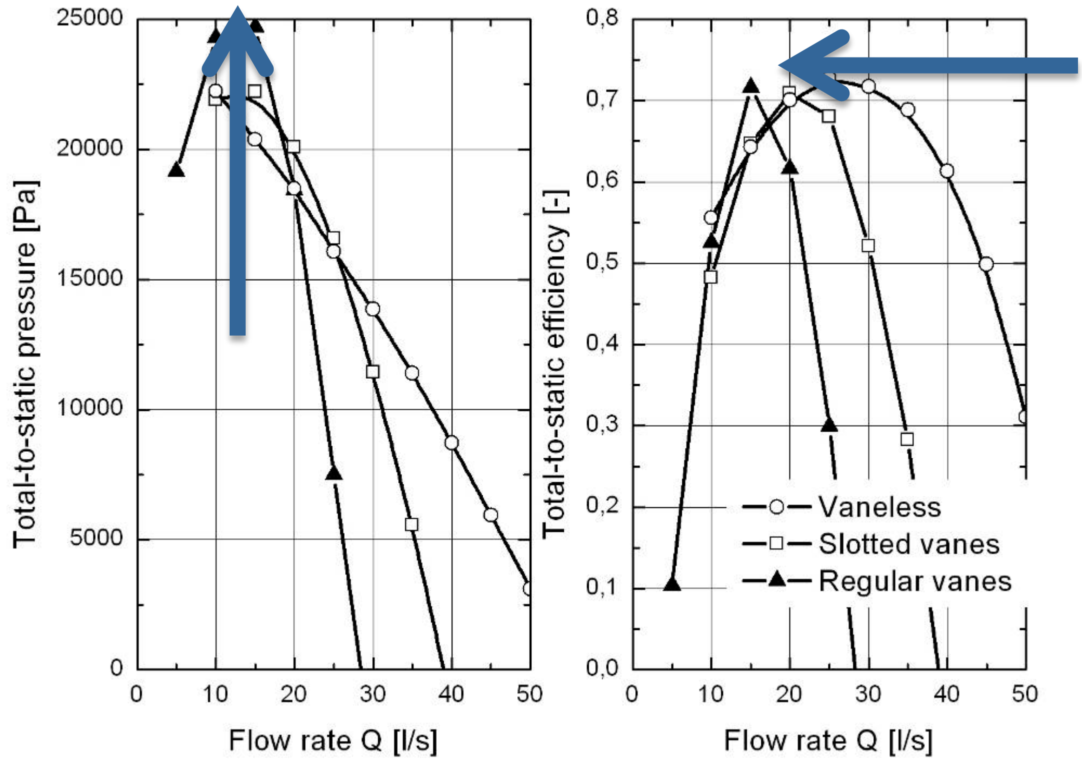
With the slots, higher flow rates are possible without choking the diffuser. In practice, instead of a single slot per blade, it has proved favourable to have four slots distributed in the inlet region of the diffuser blades.



Numerical verification



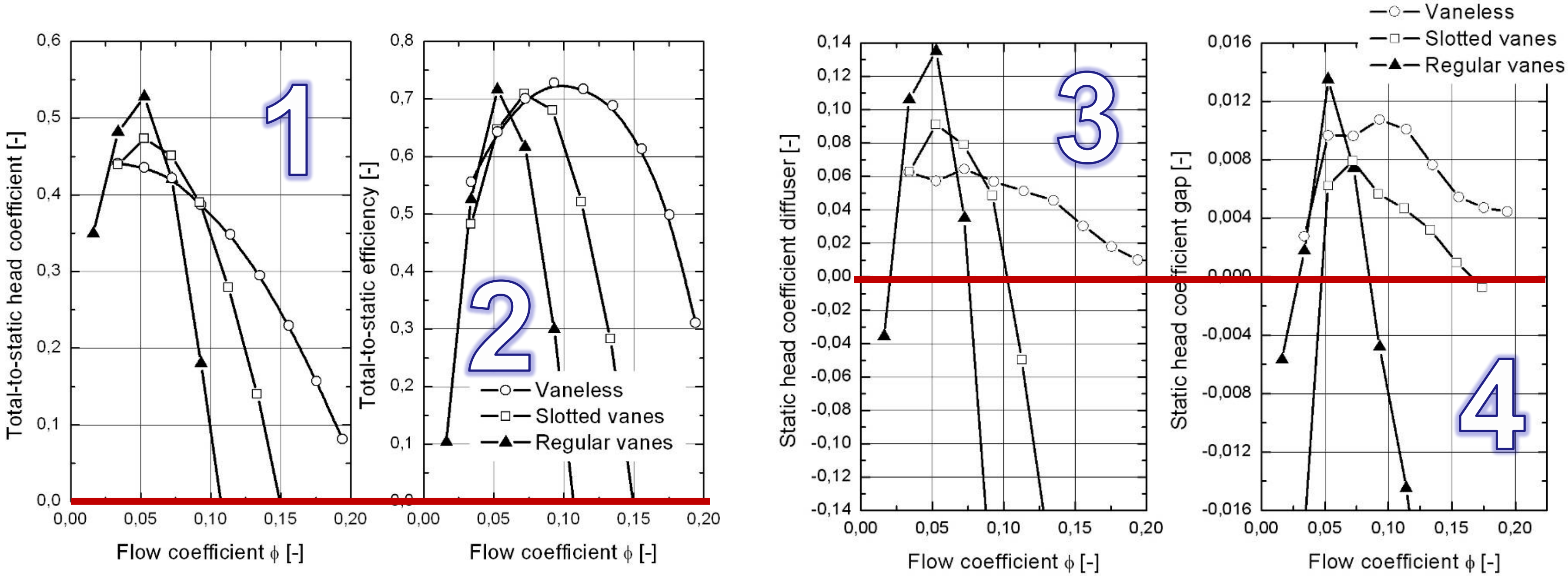
Numerical results



	Pressure	*Flow rate*	*Efficiency maximum*
vaneless	low	high	at high flow rates
slotted vanes	medium	medium	at medium flow rates
regular vanes	high	low	at low flow rates

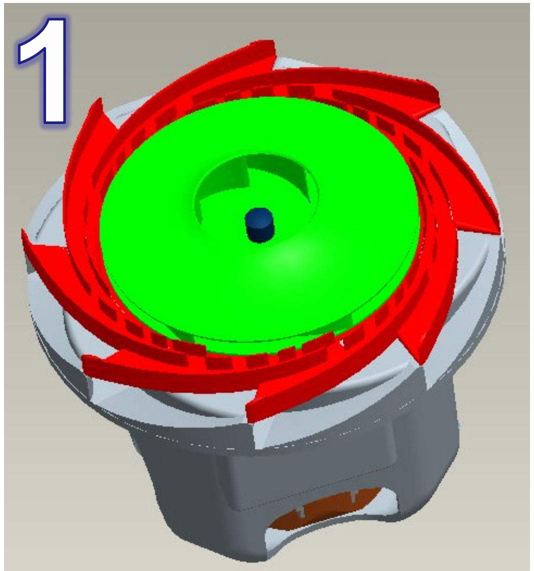
Gap losses

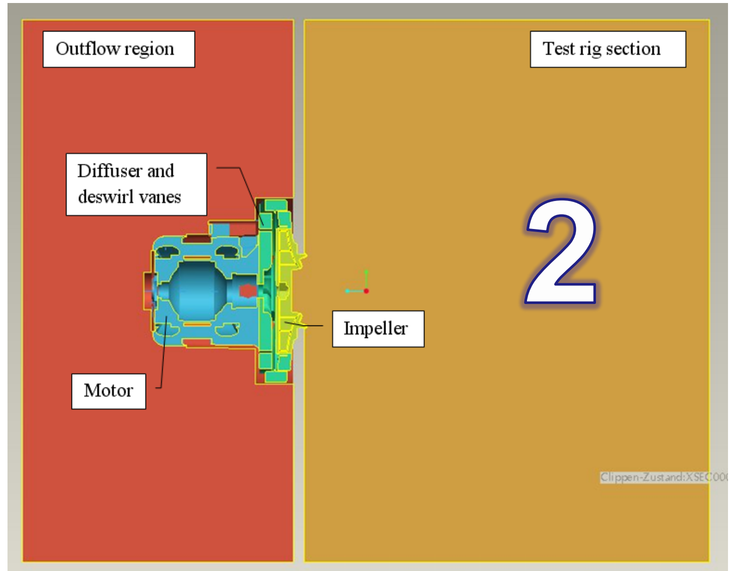
Impeller & Diffuser Diffuser Gap



1. Static head coefficient (SHC) in the diffuser only (3)
2. SHC in the gap between the impeller and the diffuser (4).
3. The vaneless diffuser always has positive SHC in the diffuser and in the gap (3 & 4)
4. The vaned diffusers have the highest values of SHC at the design point at a flow coefficient of 0.05
5. The regular vaned diffuser (no slots) being the best at this point
6. The losses in the gap are the reason why the maximum efficiency of the IDU with vaned diffusers, regular and slotted, are not higher then the IDU with the vaneless diffuser.

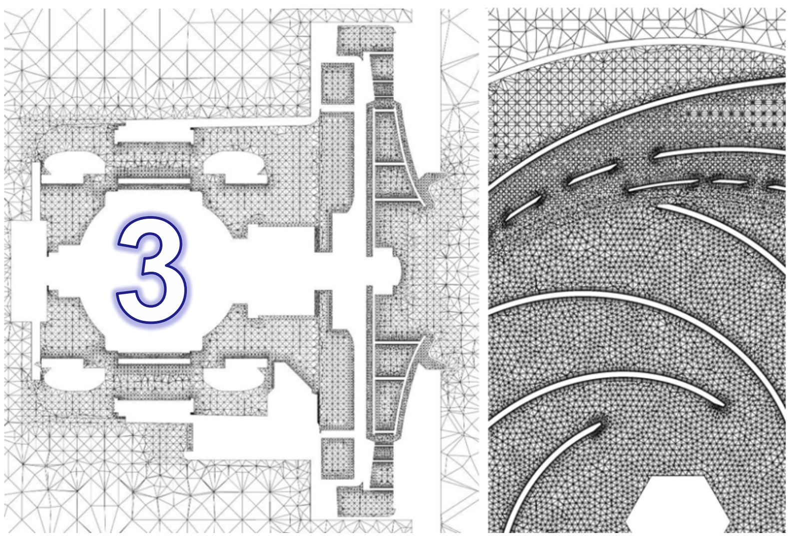
Full fan CFD simulation





CAD Model:
1) Fan
2) Full flow domain

Turbulence Model:
SST
Solver:
ANSYS CFX

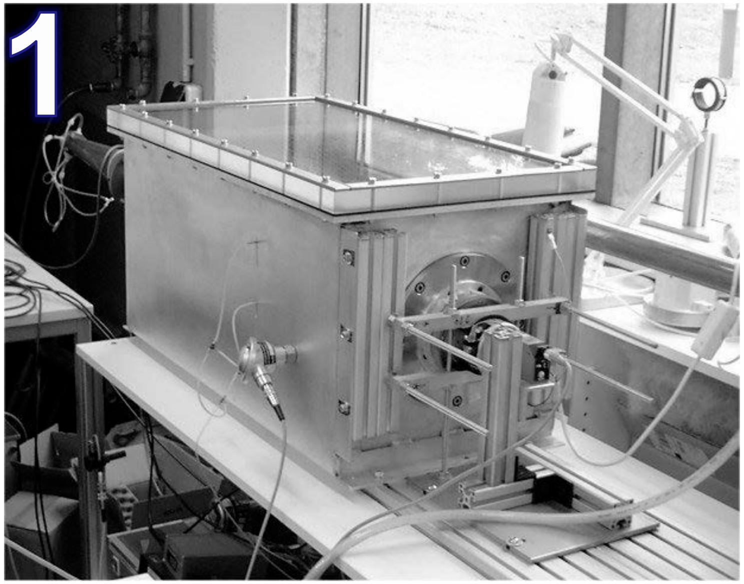


Unstructured tetrahedral grid:
3) Fan and impeller-diffuser unit
4) Size

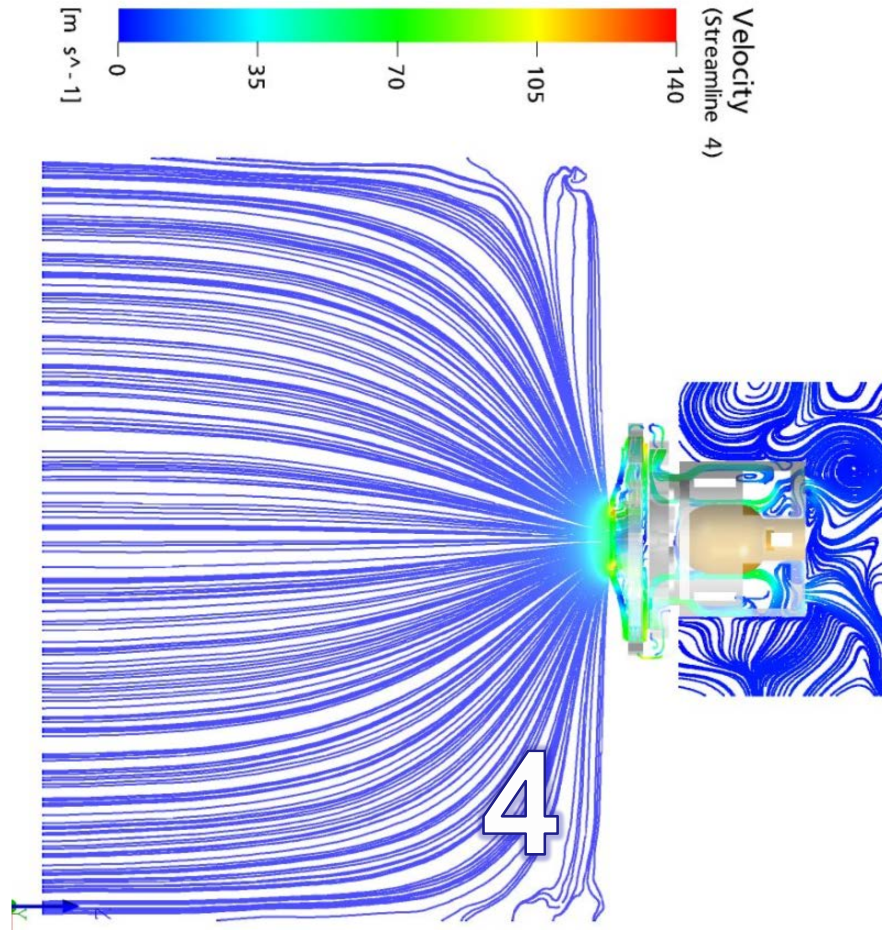
Domain	Elements
Diffuser and deswirl vanes	2,611,793
Impeller	1,625,083
Test rig section	219,683
Motor	1,857,924
Outflow region	298,431
All domains	6,612,914

4

Measurement and CFD

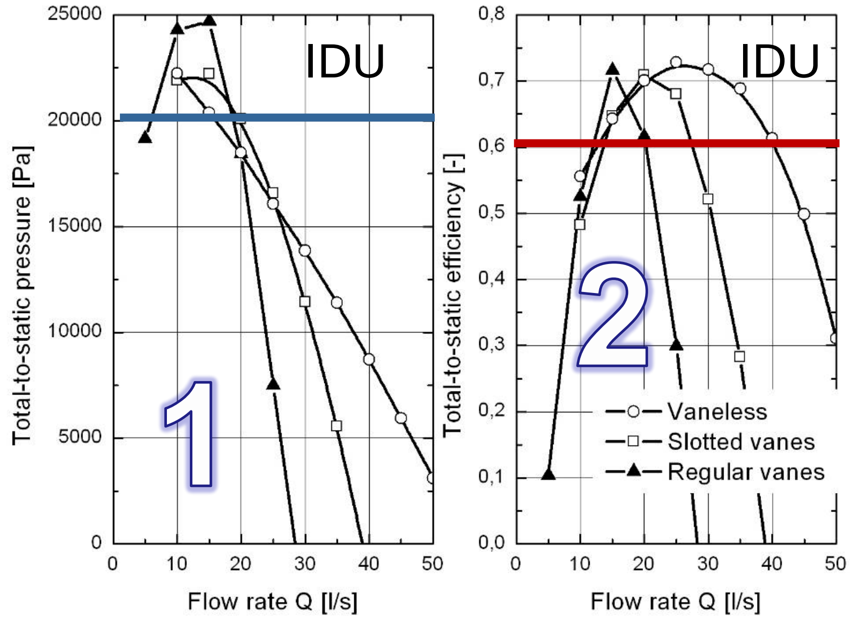


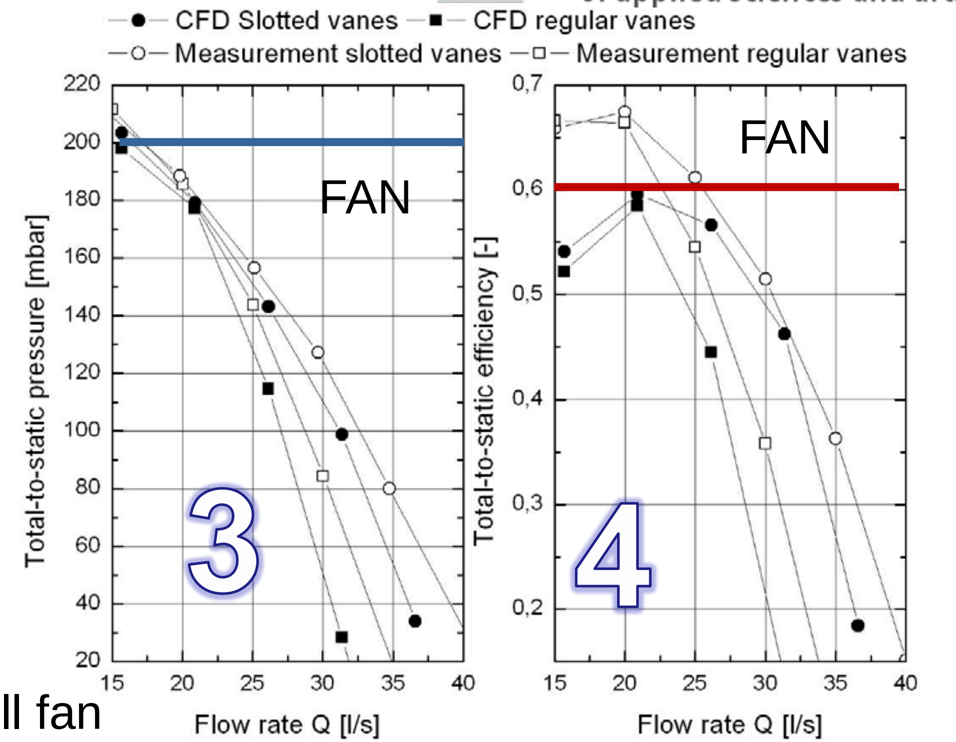
1) Test rig according to DIN 24 163
2) Prototype with slotted diffuser
3) Prototype with regular diffuser
4) Full CFD simulation



Measurement and CFD: Results and conclusion

COBURG UNIVERSITY
of applied sciences and arts





1+2) CFD results of IDU only

3+4) CFD and measuerement results of full fan

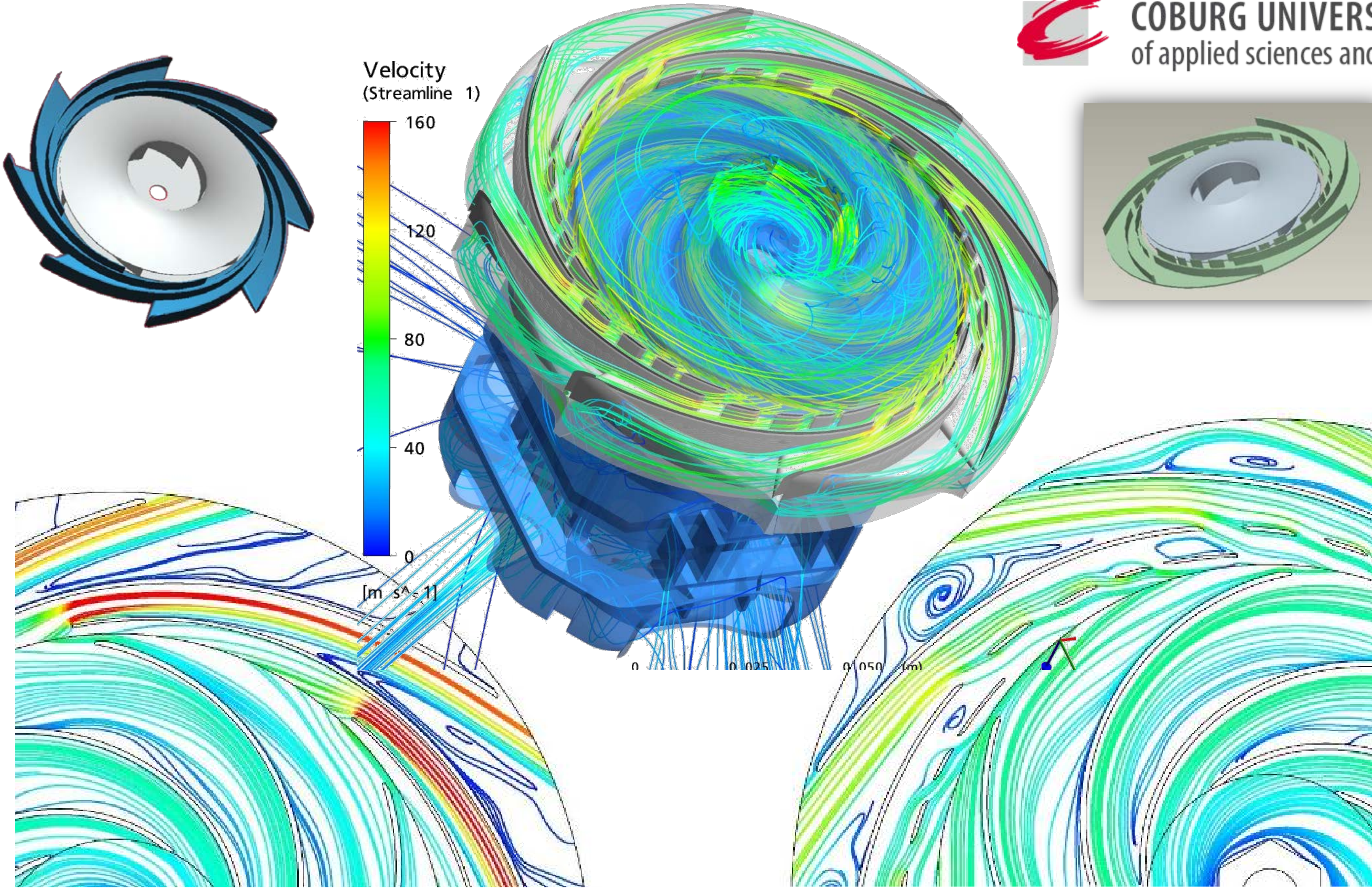
<u>Analysis:</u>

a) Decrease in maximum efficiency and pressure from IDU to Fan due to losses in the motor

b) Also in the full fan the slotted diffuser leads to higher flow rates than the regular type.

c) The maximum pressure is almost the same for the regular and slotted diffusers, which is also confirmed by the measurements

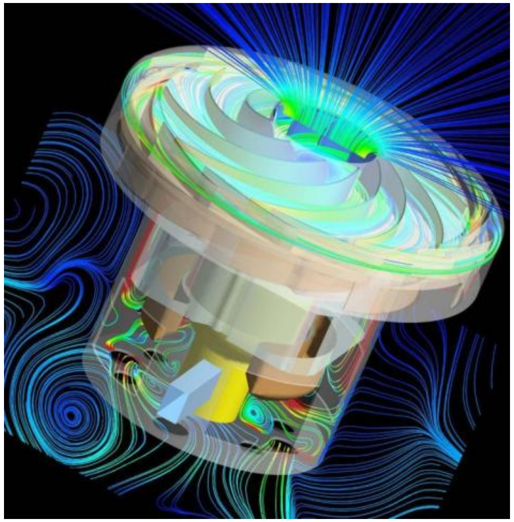
d) Hence, in the full fan, the slotted diffuser proves to be better than the regular one



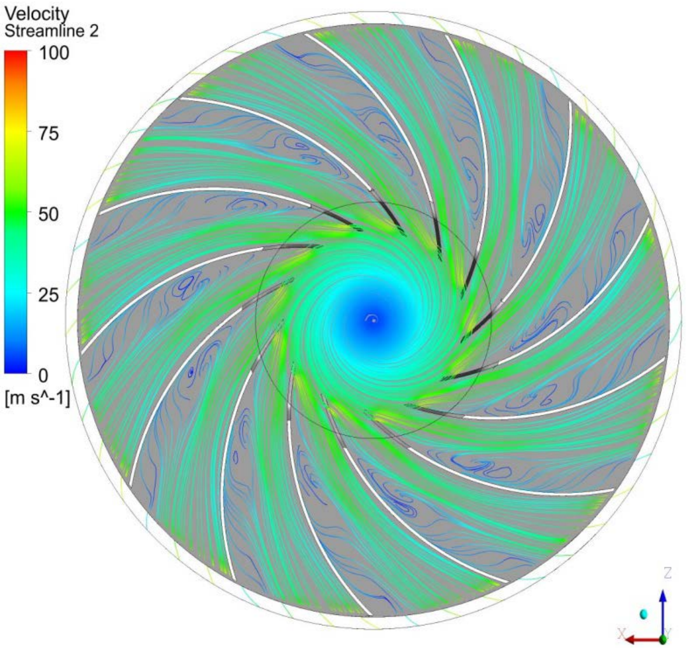
Any questions to this topic?

CONTENTS

- NEED FOR TFD, CFD AND EFD
- CASE STUDY I:
  SLOTTED DIFFUSER
- CASE STUDY II:
  TORQUE-SPEED CHARACTERISTIC
- CASE STUDY III:
  COMPACT TEST RIG DESIGN




CASE STUDY II:
A DESIGN METHOD OF RADIAL FANS CONSIDERING THE TORQUE-SPEED-CHARACTERISTIC OF THE MOTOR

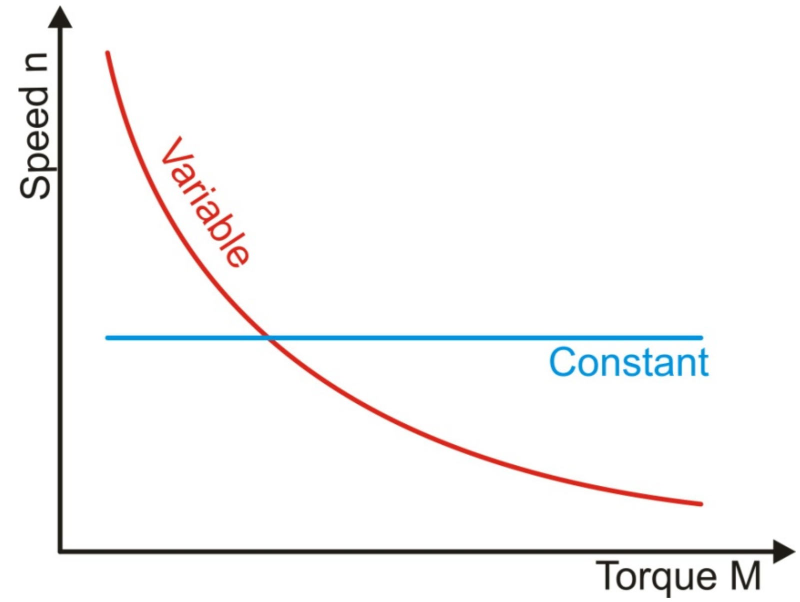


Torque-Speed-Characteristic

- The most common design case is the one with constant speed. In that case, one assigns the corresponding value to the speed n, hence the speed no longer matters in the further design procedure: it is given and it is constant.
- However, in many cases the speed is not constant, since it is governed by the torque–speed characteristic of the driving motor. In such a case it is necessary to consider this characteristic already at the design stage.

$$n = \frac{k_1}{\sqrt{M_{drive}}} + k_2$$

$$M_{drive} = \left(\frac{k_1}{n - k_2} \right)^2$$



Variable speed design

For a general drive and hence torque–speed characteristic, one can write

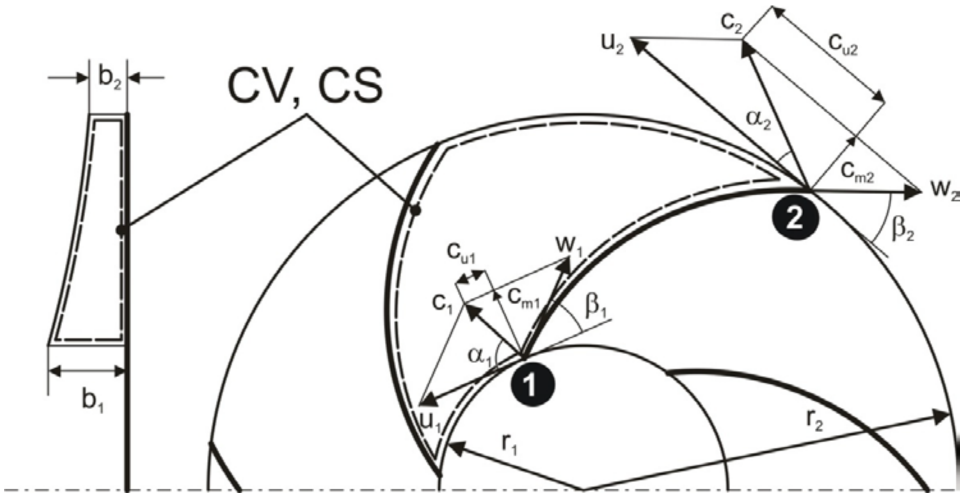
$$\Delta p_{t12} = \rho \left(u_2 \, c_{u2} - u_1 \, c_{u1} \right) = \rho \, u_2 \, c_{u2}$$

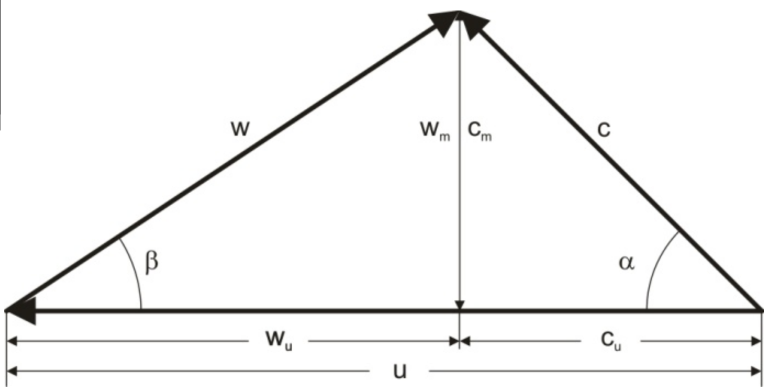
$$P_{shaft,impeller} = Q \, \Delta p_{t12}$$

$$P_{Shaft,impeller} = \rho \, Q \underbrace{\pi \, d_2 \, n}_{u_2} \left[\underbrace{\pi \, d_2 \, n - \frac{Q}{\pi \, d_2 \, b_2 \tan ß_2}}_{c_{u2}} \right]$$

$$M_{drive} = f(n)$$

$$f(n) = \frac{1}{2} \rho \, Q \, d_2 \left(\pi \, d_2 \, n - \frac{Q}{\pi \, d_2 \, b_2 \tan ß_2} \right)$$





Variable speed design

COBURG UNIVERSITY of applied sciences and arts

Torque-Speed

Speed-Flow rate

System

Motor $f(n) = A_{drive} + B_{drive}\, n$

Impeller $f(n) = \frac{1}{2} \rho Q d_2 \left(\pi d_2 n - \frac{Q}{\pi d_2 b_2 \tan ß_2} \right)$

$$n = \frac{2\ A_{drive} + \dfrac{\rho Q^2}{\pi b_2 \tan ß_2}}{\left(-2B_{drive} + \pi \rho d_2^2 Q\right)}$$

System

Motor $M_{motor} = f(n)$

Impeller $M_{impeller} = g(Q,n)$

$$\left.\begin{matrix} M_{motor} = f(n) \\ M_{impeller} = g(Q,n) \end{matrix}\right\} \Rightarrow n = h(Q) \Rightarrow \begin{cases} M_{impeller} = M_{impeller}(Q) \\ \Delta p_{t-s} = \Delta p_{t-s}(Q) \\ \eta_{t-s} = \eta_{t-s}(Q) \end{cases}$$

Variable speed design

COBURG UNIVERSITY
of applied sciences and arts

$$n = \frac{2\ A_{drive} + \dfrac{\rho Q^2}{\pi\, b_2 \tan\ ß_2}}{\left(-2B_{drive} + \pi\,\rho\, d_2^2\, Q\right)}$$

$$M = \frac{1}{2}\rho\, Q\, d_2 \left(\pi\, d_2\, n - \frac{Q}{\pi\, d_2\, b_2 \tan\ ß_2}\right)$$

$$\Delta p_{t-s} = \frac{\rho}{2}\left[\left(\pi\, d_2\, n\right)^2 - \left(\frac{Q}{\pi\, d_2\, b_2 \sin(ß_2)}\right)^2\right]$$

$$\eta_{t-s} = \frac{1}{2}\frac{\left[\left(\pi\, d_2\, n\right)^2 - \left(\dfrac{Q}{\pi\, d_2\, b_2 \sin\ ß_2}\right)^2\right]}{\pi\, d_2\, n\left(u_2 - \dfrac{Q}{\pi\, d_2\, b_2 \tan\ ß_2}\right)}$$

Speed as a function of flow rate $n = h(Q)$

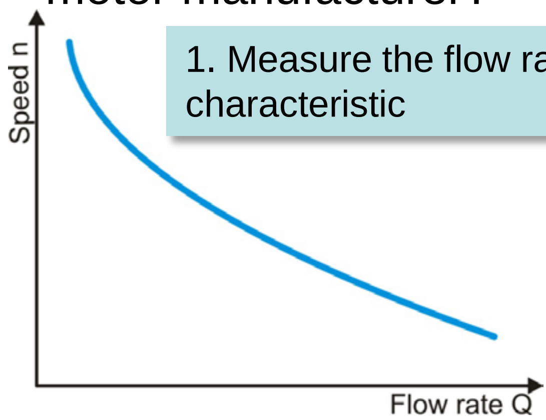
Torque, pressure and efficiency as a function of flow rate

$$\Rightarrow \begin{cases} M_{impeller} = M_{impeller}(Q) \\ \Delta p_{t-s} = \Delta p_{t-s}(Q) \\ \eta_{t-s} = \eta_{t-s}(Q) \end{cases}$$

Motor torque-speed-characteristic

COBURG UNIVERSITY
of applied sciences and arts

How to proceed if the torque-speed-characteristic is not available from the motor manufacturer?

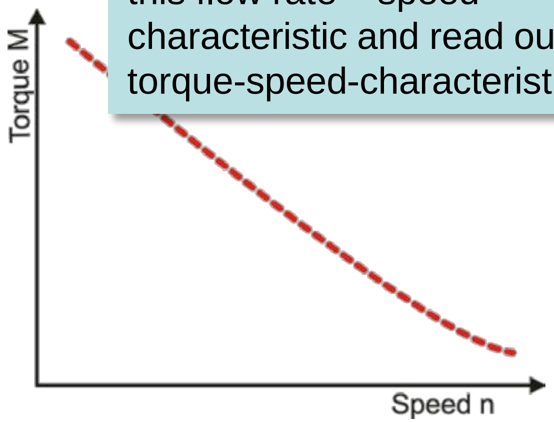


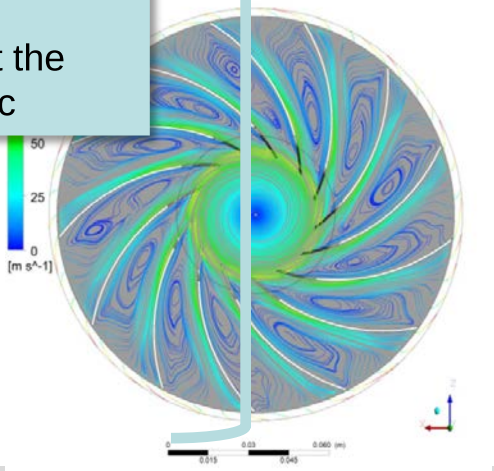
1. Measure the flow rate – speed characteristic

Flow rate

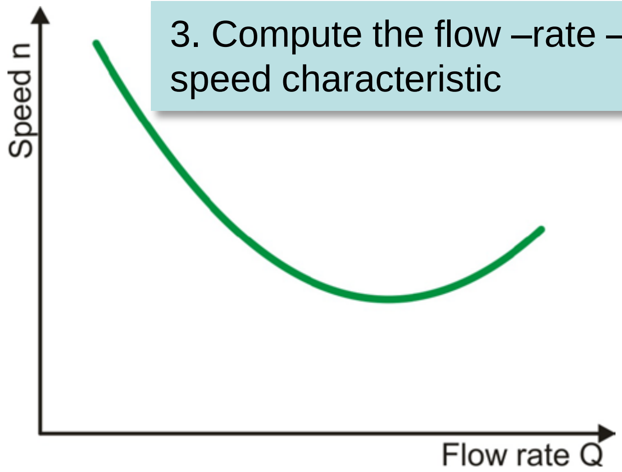
Speed

2. Setup a CFD computation at this flow rate – speed characteristic and read out the torque-speed-characteristic





3. Compute the flow –rate – speed characteristic



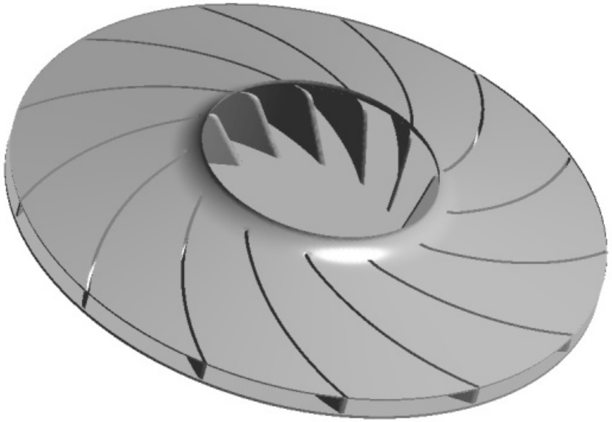
Design Case

COBURG UNIVERSITY
of applied sciences and arts

Aims of the project:

1. Keep or improove the hydraulic power
2. Increase the rotating speed due to the cooling fan at the other end of the shaft
3. Improove the hydraulic efficiency

→ Starting point is the torque-speed-characteristic of the motor



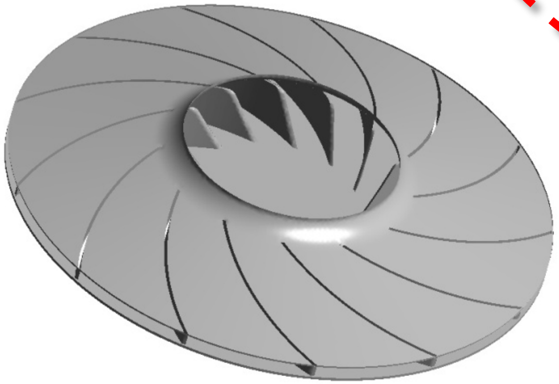
Sheet metal impeller

Design Case

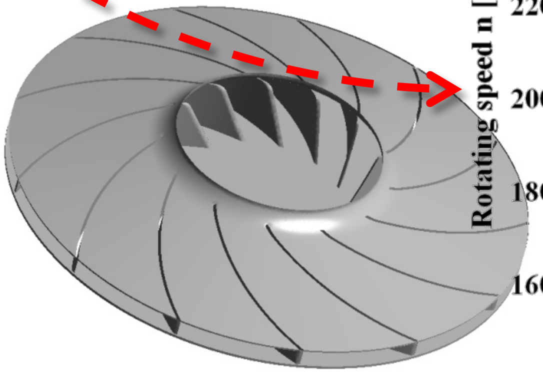
COBURG UNIVERSITY
of applied sciences and arts

Blende (mm)	n (min^{-1})	p korrigiert (mmWS)	V (ls^{-1})	i (A)	P (W)	P2 (W)	η (%)
50	16580	371,0	91,49	6,96	1597	332,88	20,84
40	17351	690,8	79,90	7,05	1627	541,25	33,27
30	18819	1115,1	57,10	7,02	1618	624,42	38,59
23	20269	1423,6	37,92	6,75	1553	529,42	34,09
19	21382	1581,4	27,28	6,56	1528	423,01	27,68
16	22381	1694,1	20,02	6,43	1475	332,61	22,55
13	23266	1810,0	13,66	6,28	1440	242,47	16,84
10	24280	1918,6	8,32	6,13	1406	156,58	11,14
6,5	25367	1992,4	3,58	5,95	1388	70,01	5,04
0	25822	2116,4	0,00	5,86	1360	0,00	0,00

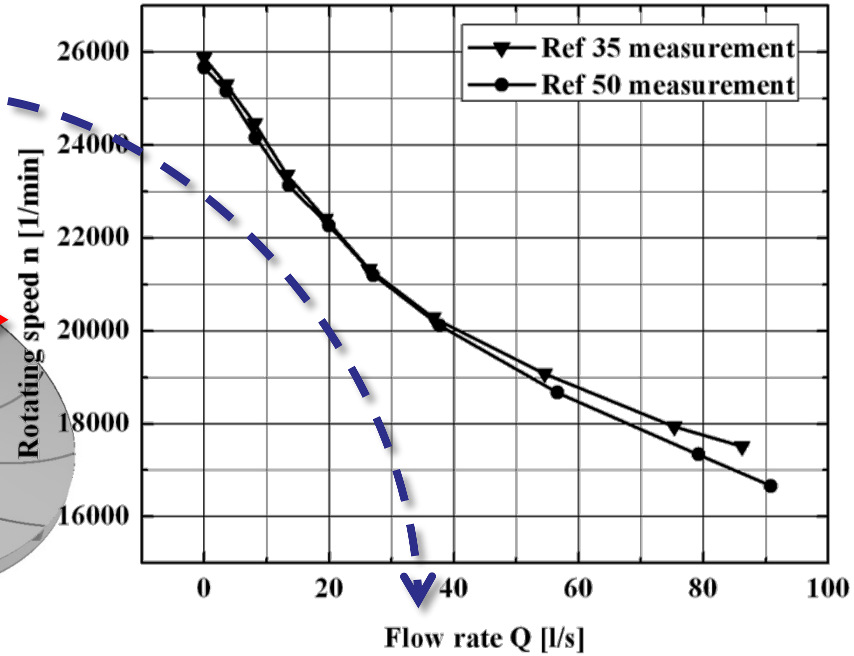
1. Measure the flow rate – speed characteristic



Ref 35

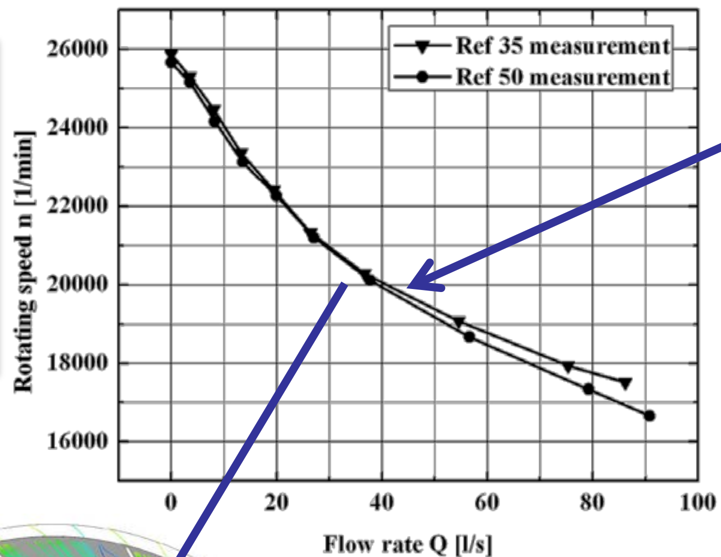


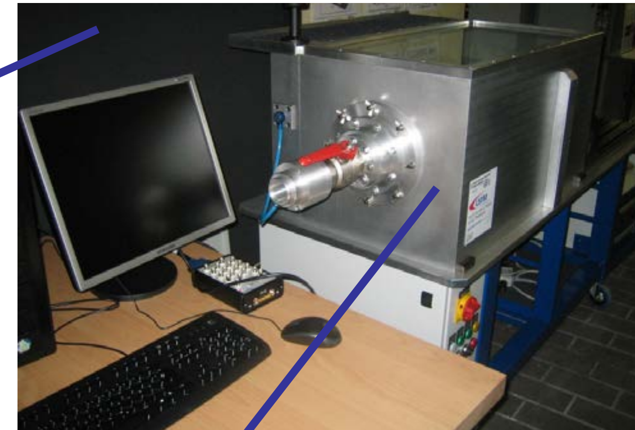
Ref 50

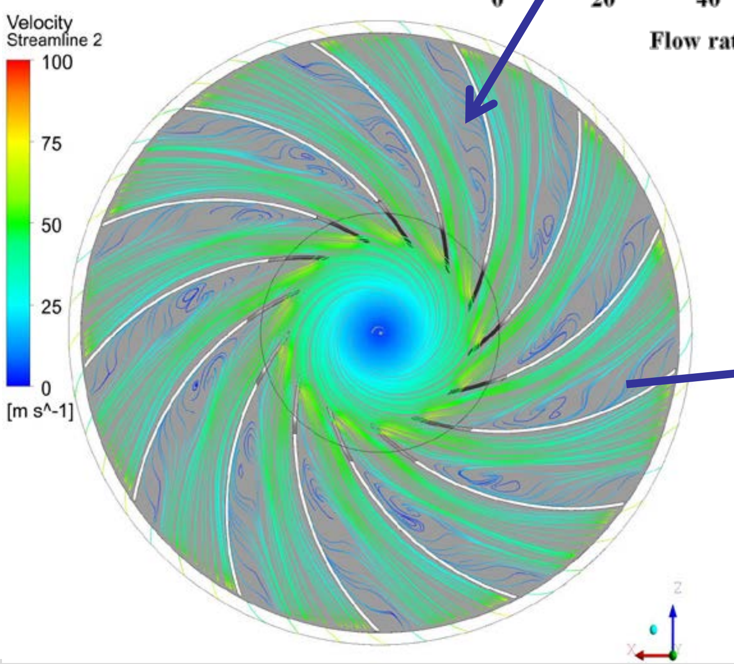


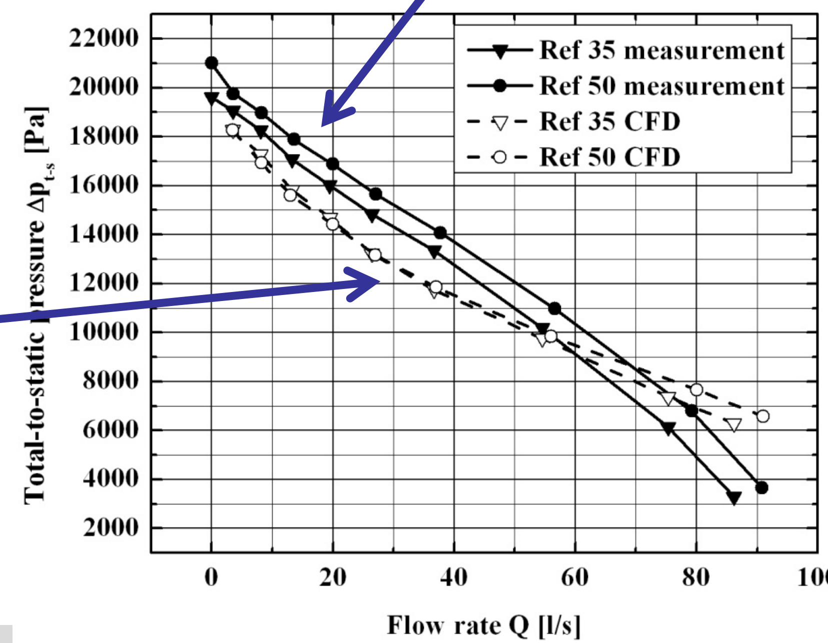
Flow rate, speed and pressure

2. Setup a CFD computation at this flow rate – speed characteristic and read out the torque-speed-characteristic



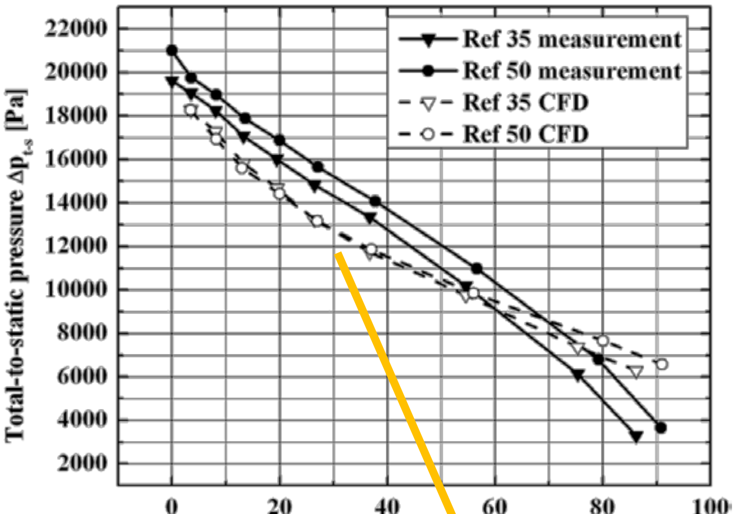


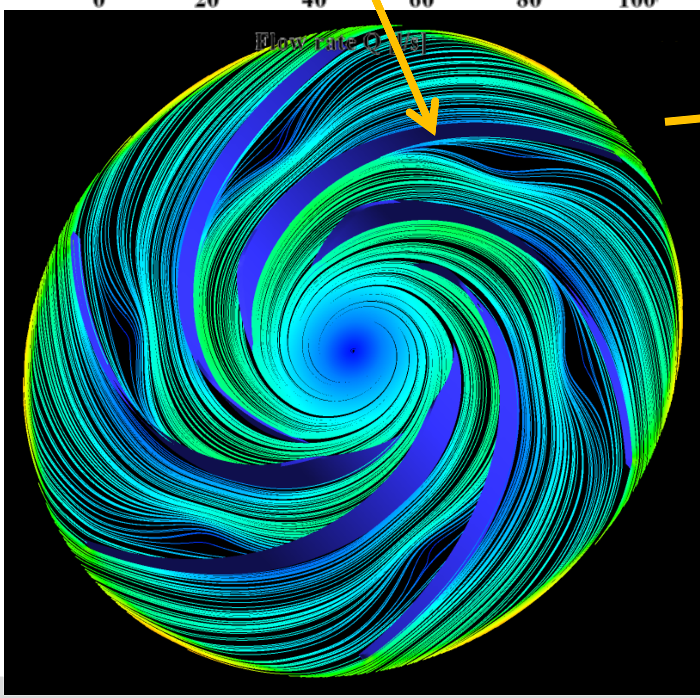




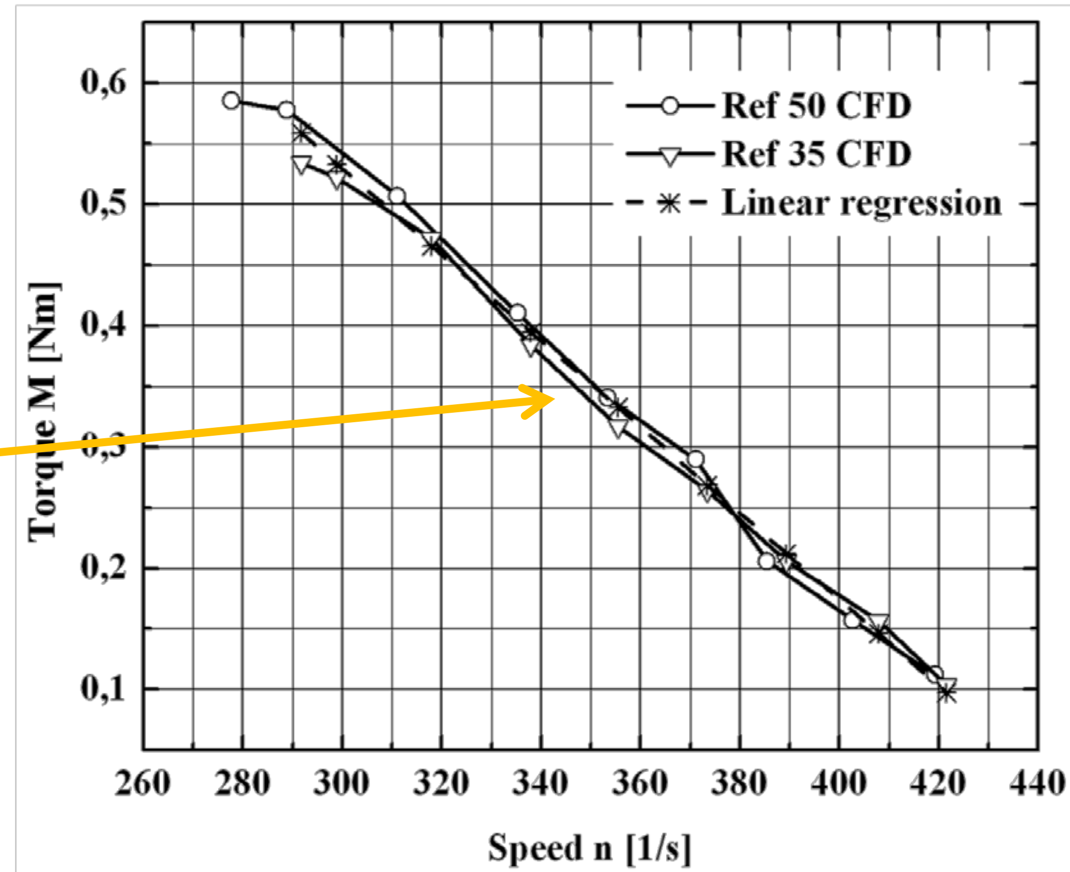
Torque-Speed-Characteristic

COBURG UNIVERSITY
of applied sciences and arts



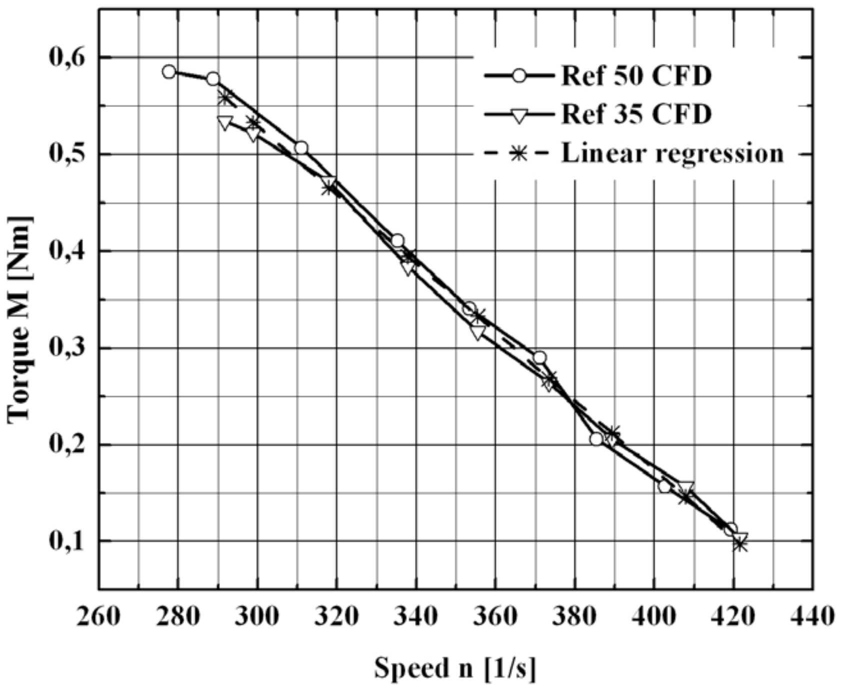


3. Compute the flow –rate – speed characteristic



Torque-speed-characteristic

Rotational Speed



$$f(n) = A_{drive} + B_{drive}\, n$$

$$n = \frac{2\, A_{drive} + \dfrac{\rho Q^2}{\pi b_2 \tan ß_2}}{\left(-2B_{drive} + \pi \rho d_2^2 Q\right)}$$

Linear regression:

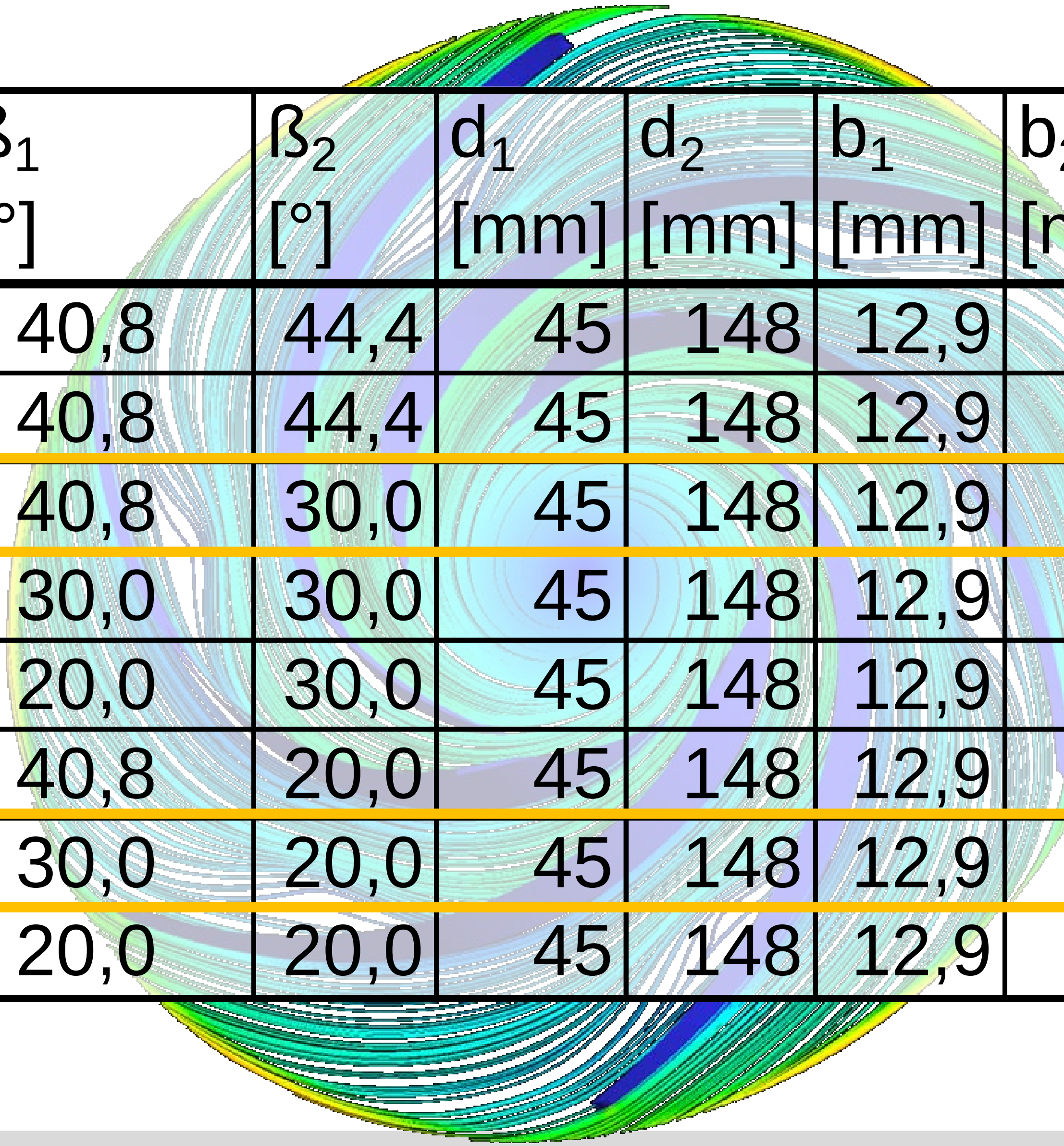
A_{drive}=1.595

B_{drive}=-0.00355

$$\boxed{n = \frac{3.19 + \dfrac{\rho Q^2}{\pi b_2 \tan ß_2}}{\left(0.0071 + \pi \rho d_2^2 Q\right)}}$$

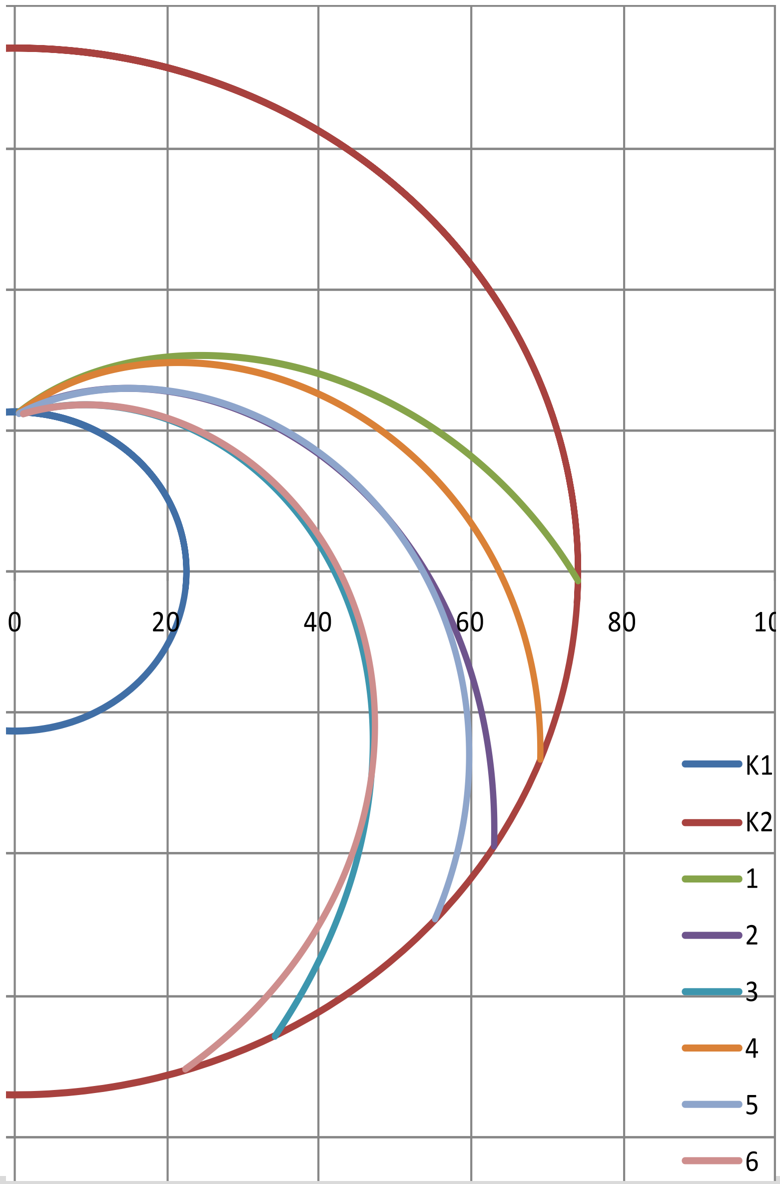
Designs

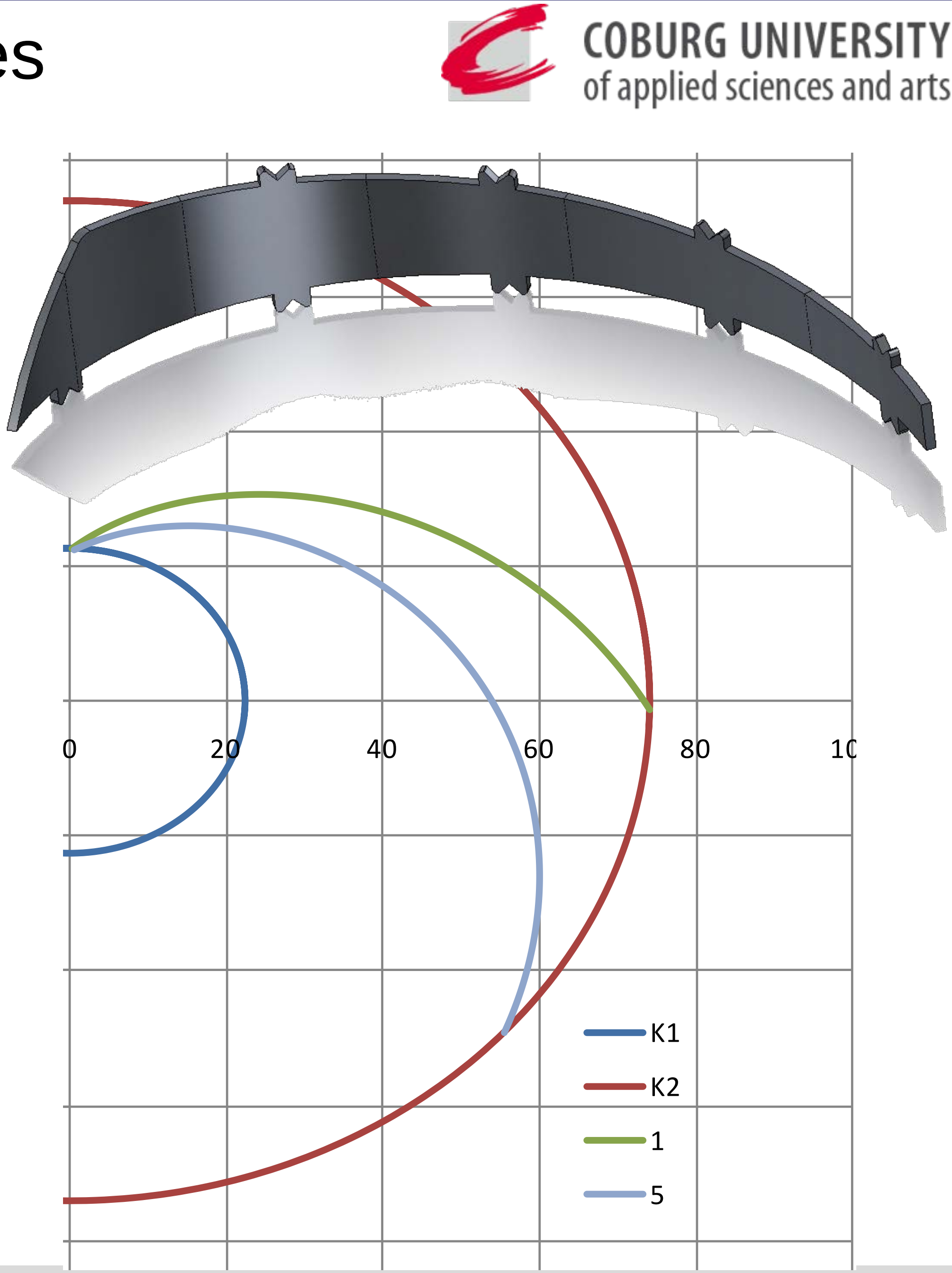
COBURG UNIVERSITY
of applied sciences and arts



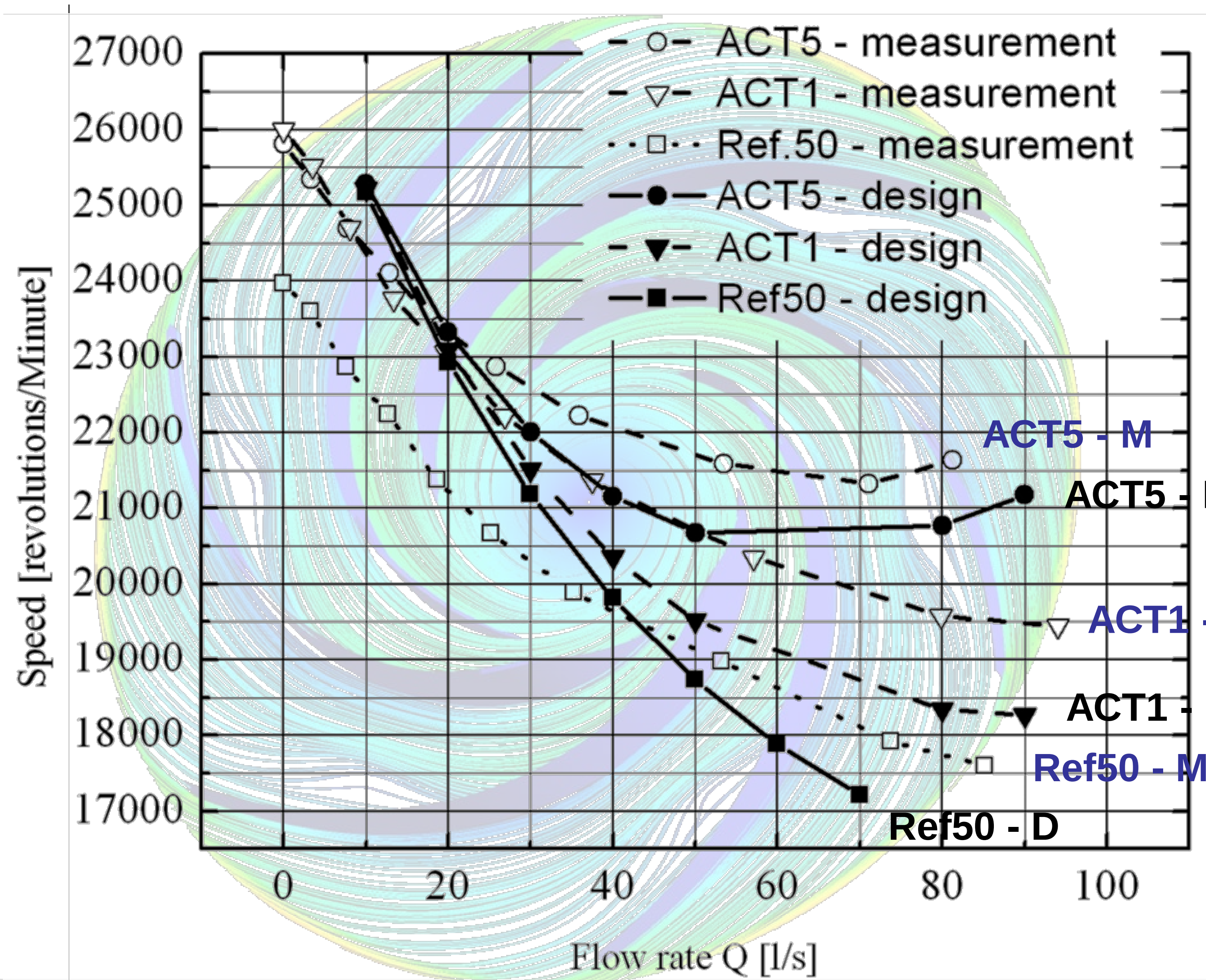
	ß$_1$ [°]	ß$_2$ [°]	d$_1$ [mm]	d$_2$ [mm]	b$_1$ [mm]	b$_2$ [mm]	θ [°]	z
Ref 50	40,8	44,4	45	148	12,9	5	60	13
Ref 35	40,8	44,4	45	148	12,9	3,5	60	13
ACT1	40,8	30,0	45	148	12,9	5	90	8
ACT2	30,0	30,0	45	148	12,9	5	120	8
ACT3	20,0	30,0	45	148	12,9	5	150	6
ACT4	40,8	20,0	45	148	12,9	5	110	7
ACT5	30,0	20,0	45	148	12,9	5	130	6
ACT6	20,0	20,0	45	148	12,9	5	160	6

Blade shapes



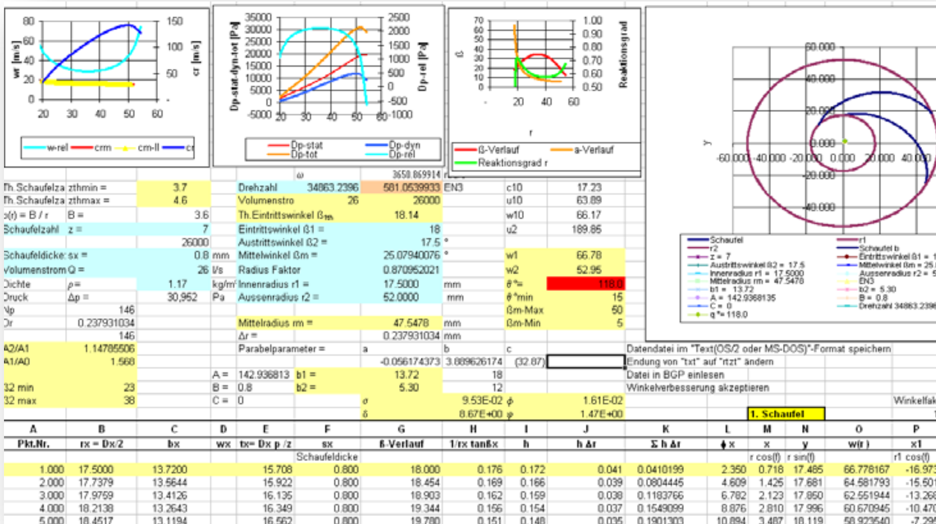


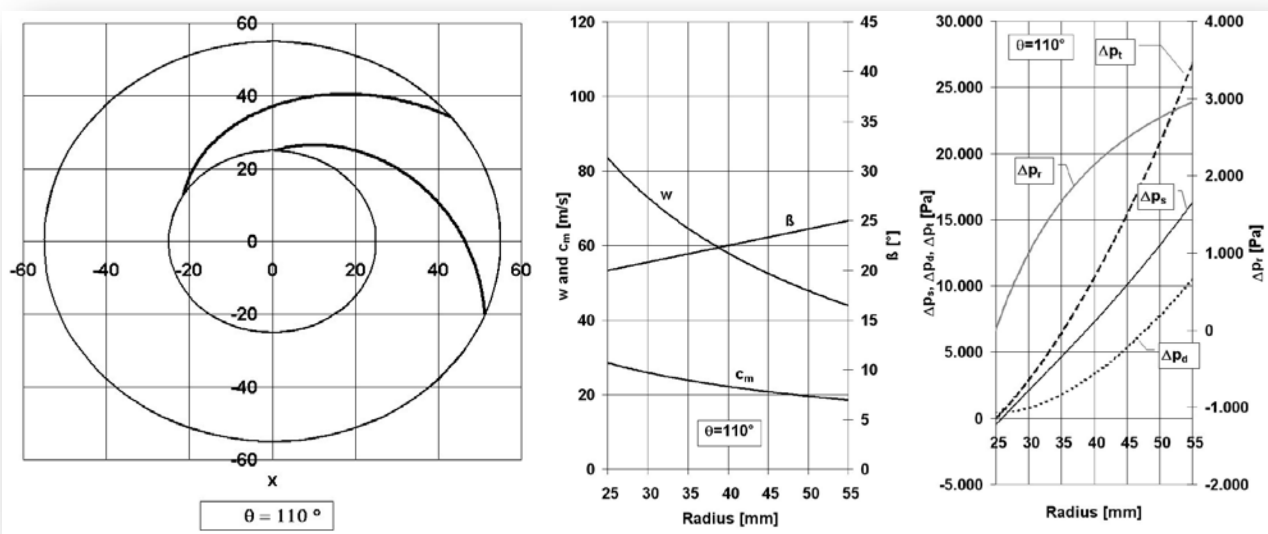
Results


ACT5 - measurement
ACT1 - measurement
Ref.50 - measurement
ACT5 - design
ACT1 - design
Ref50 - design
Speed [revolutions/Minute]
27000
26000
25000
24000
23000
22000
21000
20000
19000
18000
17000
0
20
40
60
80
100
Flow rate Q [l/s]
ACT5 - M
ACT5 - D
ACT1 - M
ACT1 - D
Ref50 - M
Ref50 - D

Blade Design Process

COBURG UNIVERSITY
of applied sciences and arts





Design software
(Microsoft Excel)

↓

Data files:
blade and
shroud

↓

CAD model
(PTC ProEngineer)

↓

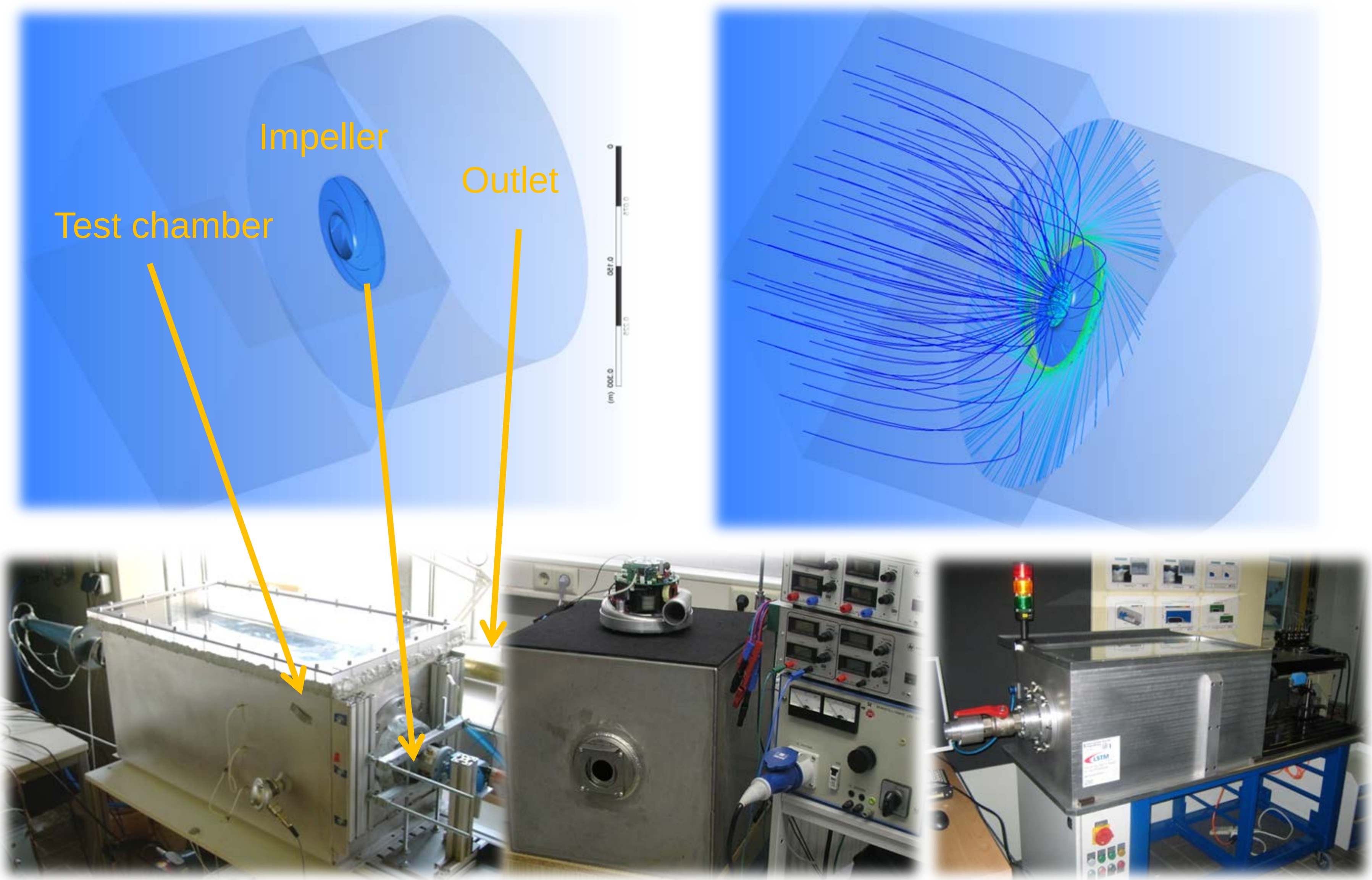
Grid
(ANSYS ICEM)

↓

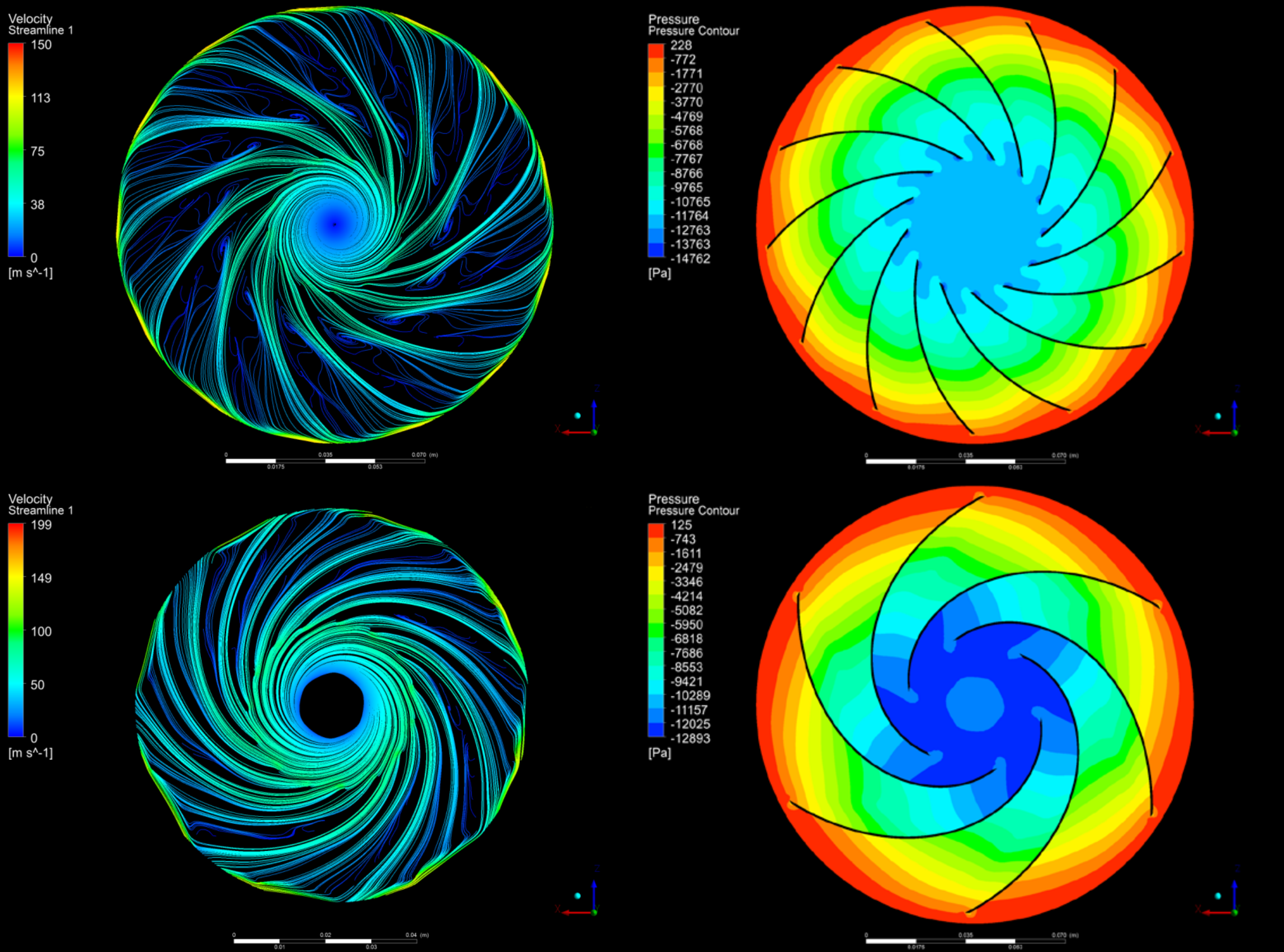
Numerical solution
(ANSYS CFX)

Flow domains

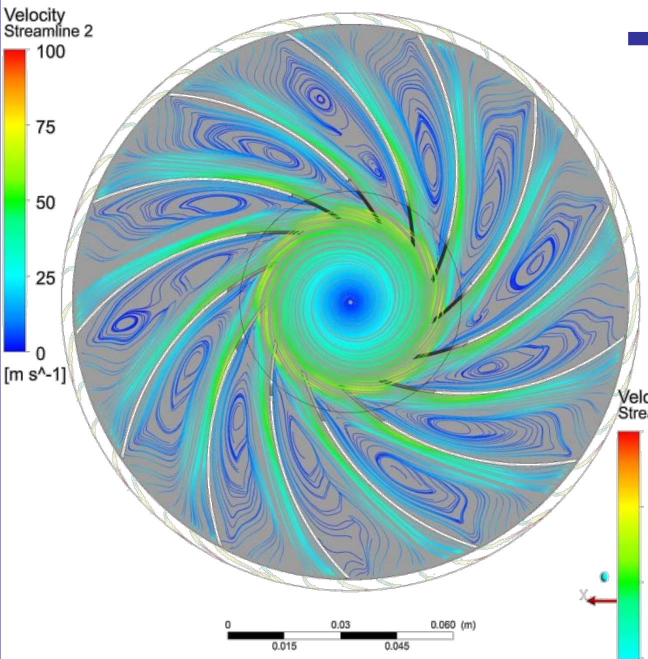
COBURG UNIVERSITY
of applied sciences and arts

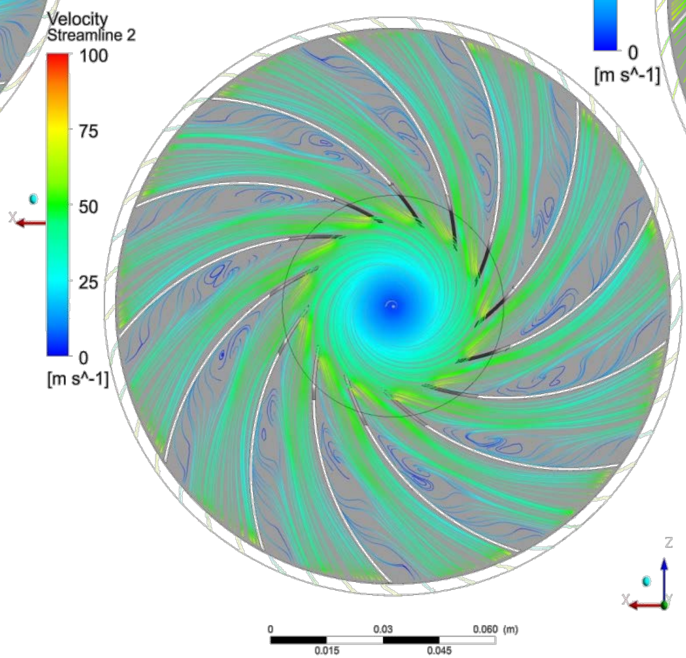


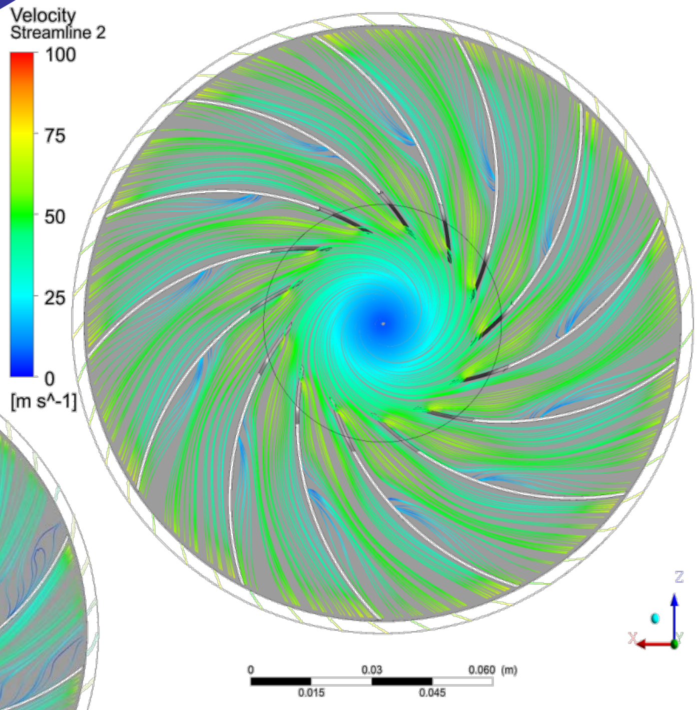
CFD


Velocity
Streamline 1
150
113
75
38
0
[m s^-1]
Pressure
Pressure Contour
228
-772
-1771
-2770
-3770
-4769
-5768
-6768
-7767
-8766
-9765
-10765
-11764
-12763
-13763
-14762
[Pa]
Velocity
Streamline 1
199
149
100
50
0
[m s^-1]
Pressure
Pressure Contour
125
-743
-1611
-2479
-3346
-4214
-5082
-5950
-6818
-7686
-8553
-9421
-10289
-11157
-12025
-12893
[Pa]

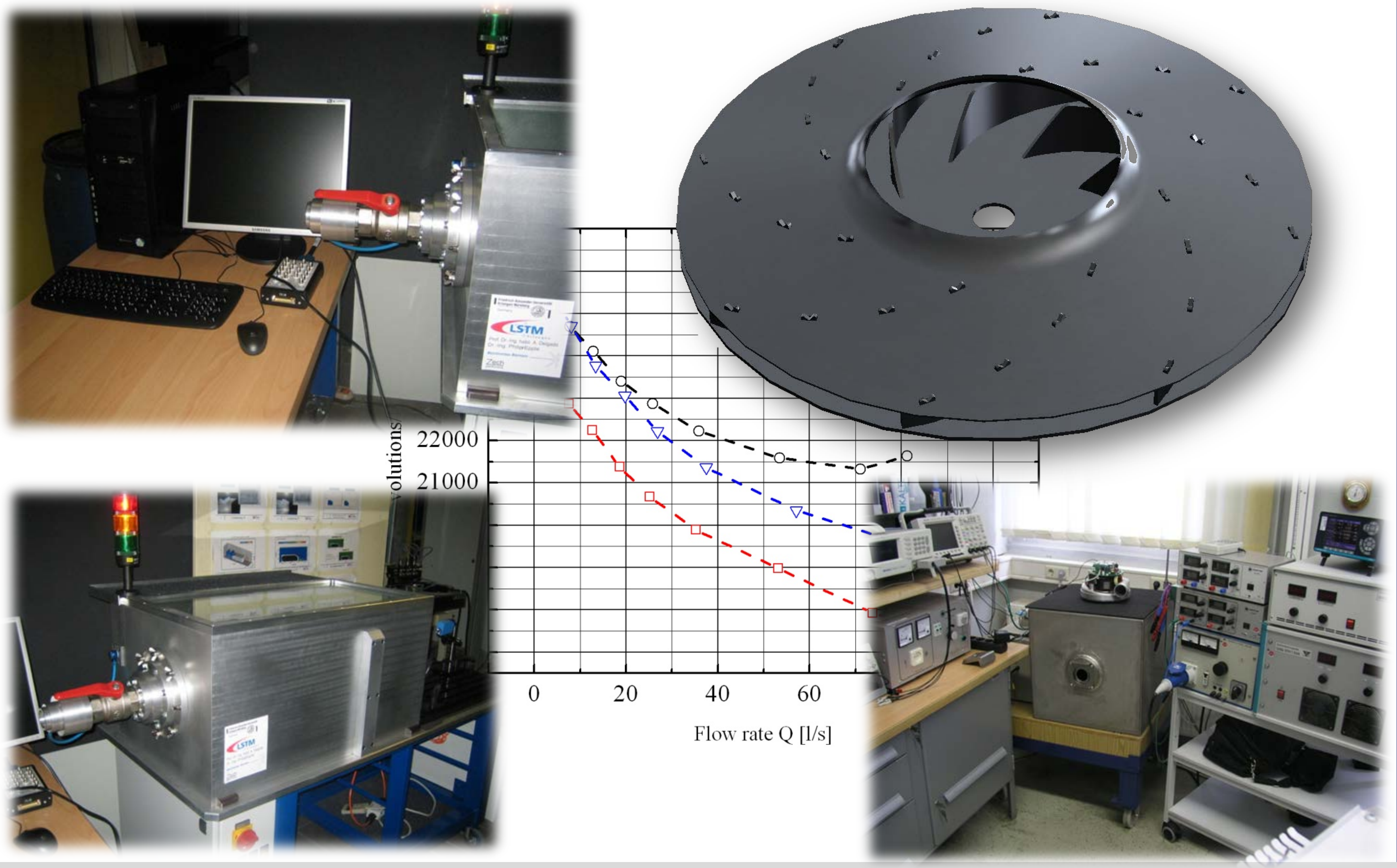
Increasing flow rate



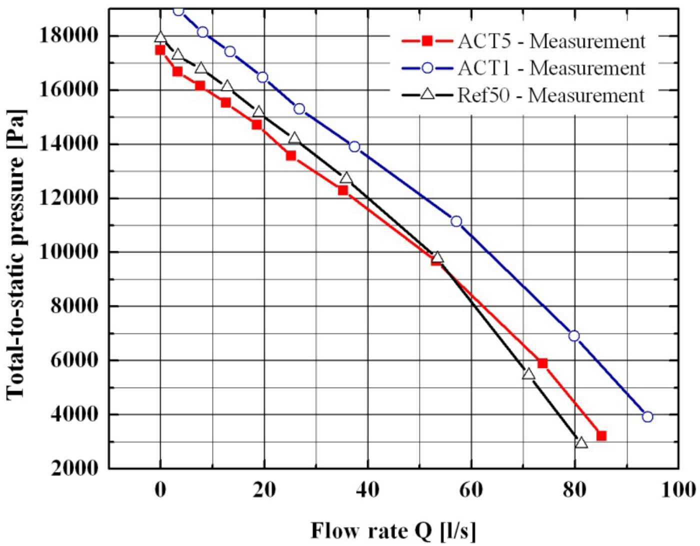


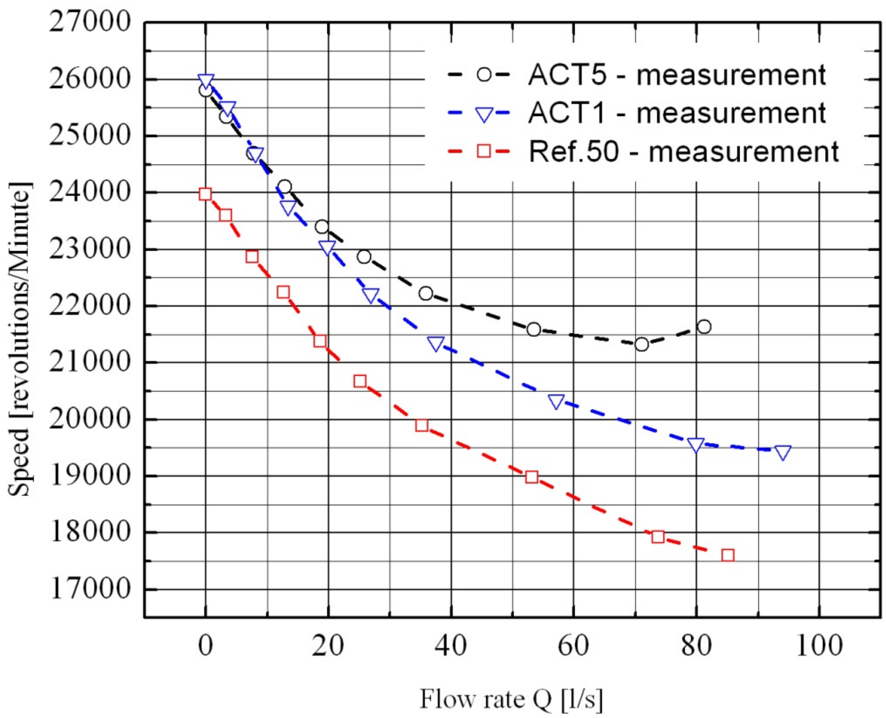


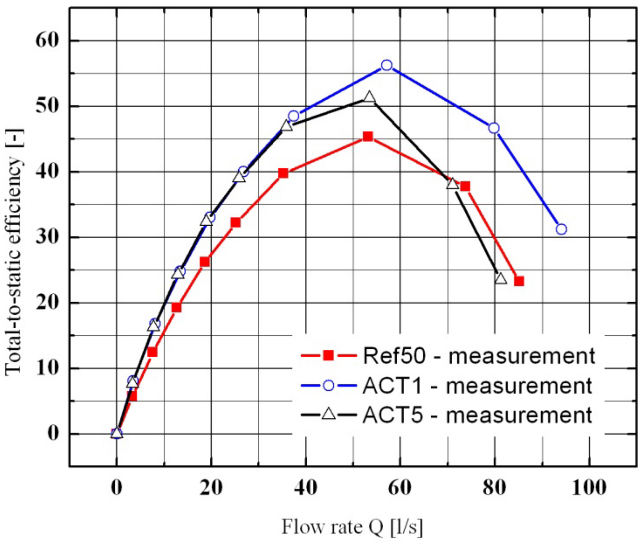
Measurements



Measured Results

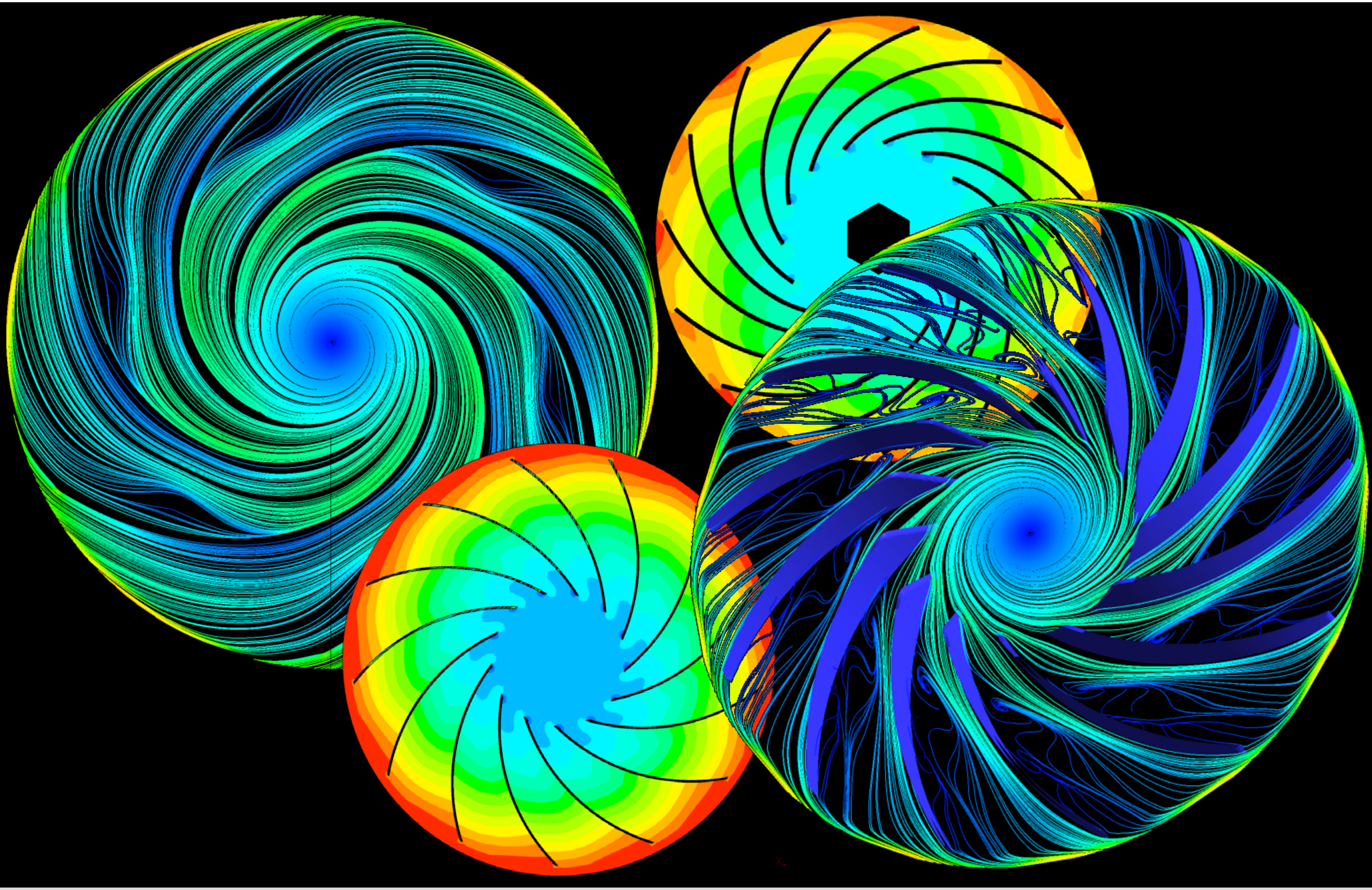






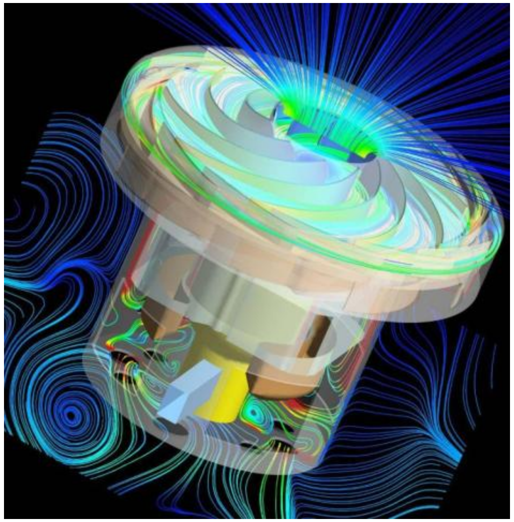
1. Higher rotational speed
2. Higher total-to-static pressure
3. Higher total-to-static efficiency

Thank you. Any questions?

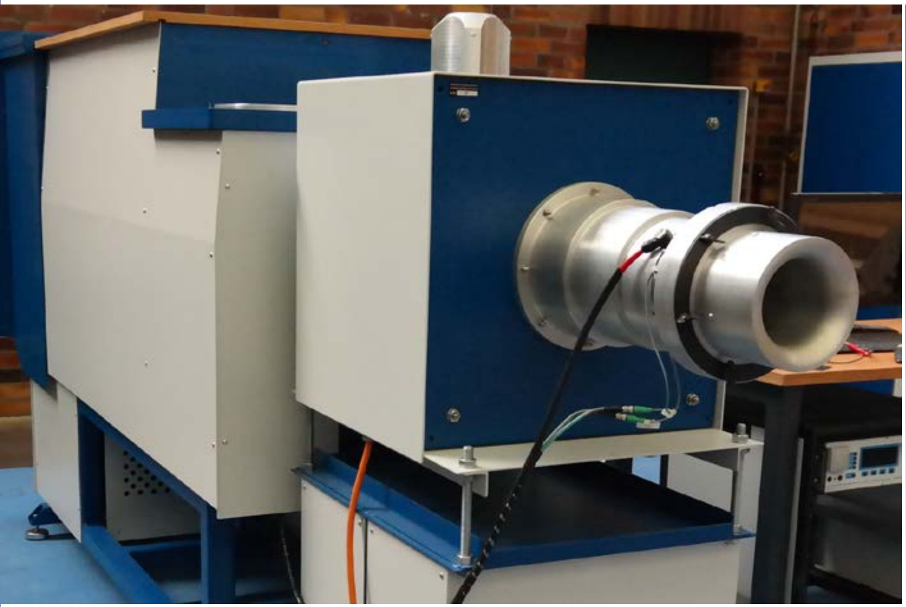


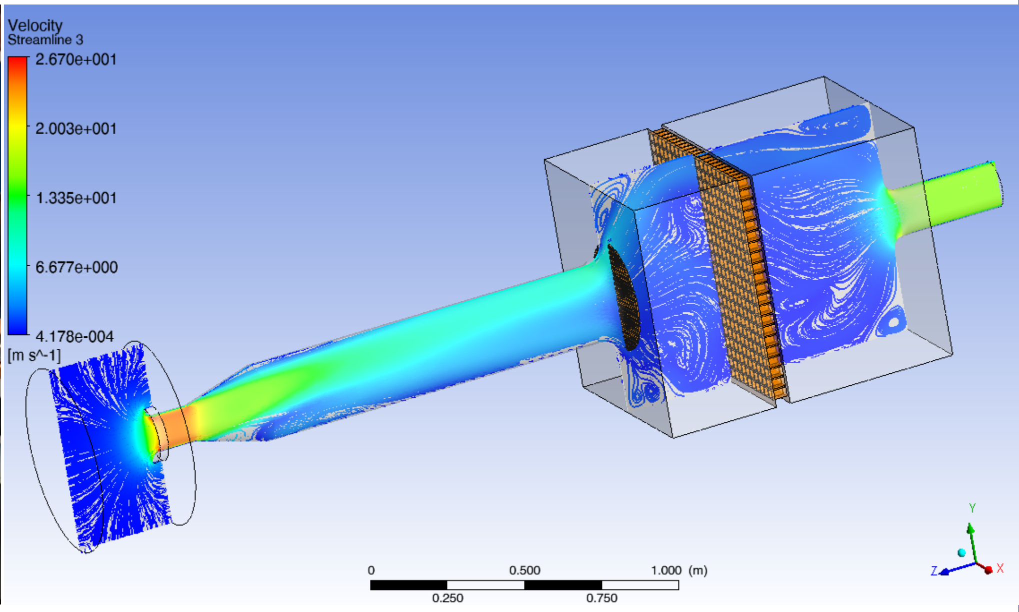
CONTENTS

- NEED FOR TFD, CFD AND EFD
- CASE STUDY I:
  SLOTTED DIFFUSER
- CASE STUDY II:
  TORQUE-SPEED CHARACTERISTIC
- CASE STUDY III:
  COMPACT TEST RIG DESIGN



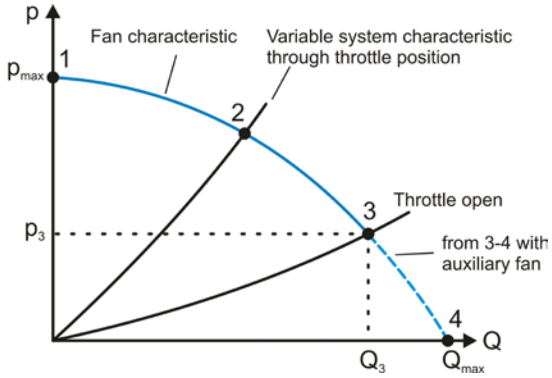
CASE STUDE III
COMPACT TEST RIG DESIGN
FOR FANS AND BLOWERS

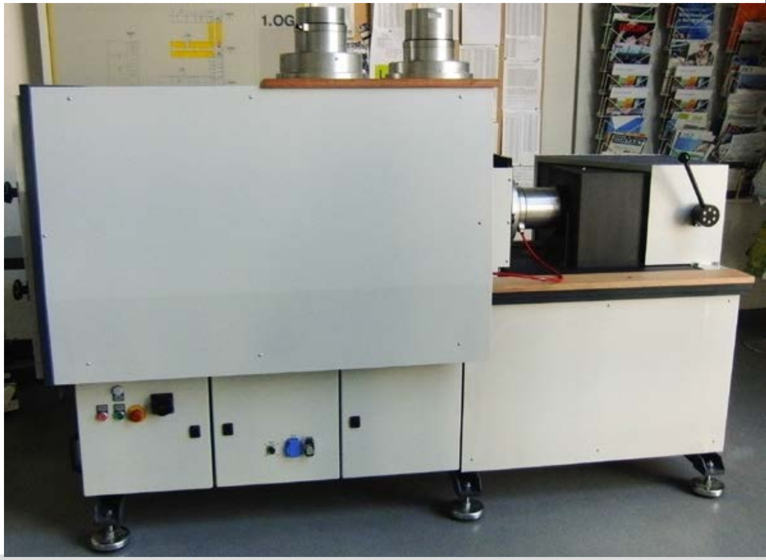




Contents

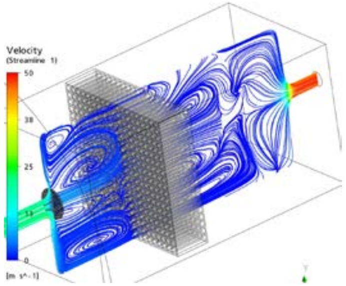
- Motivation
- Fans and blowers characteristics and their measurement
- Test rig overview
- Suction and pressure side test rigs
- Pressure side test rigs details
- Suction side test rigs details
- Conclusions

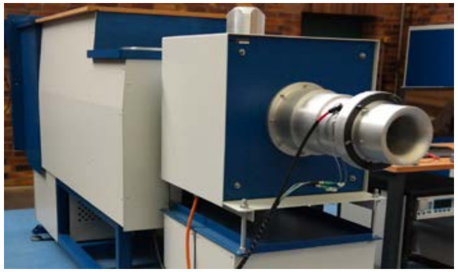
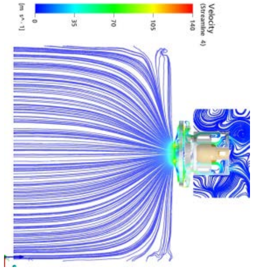




Motivation

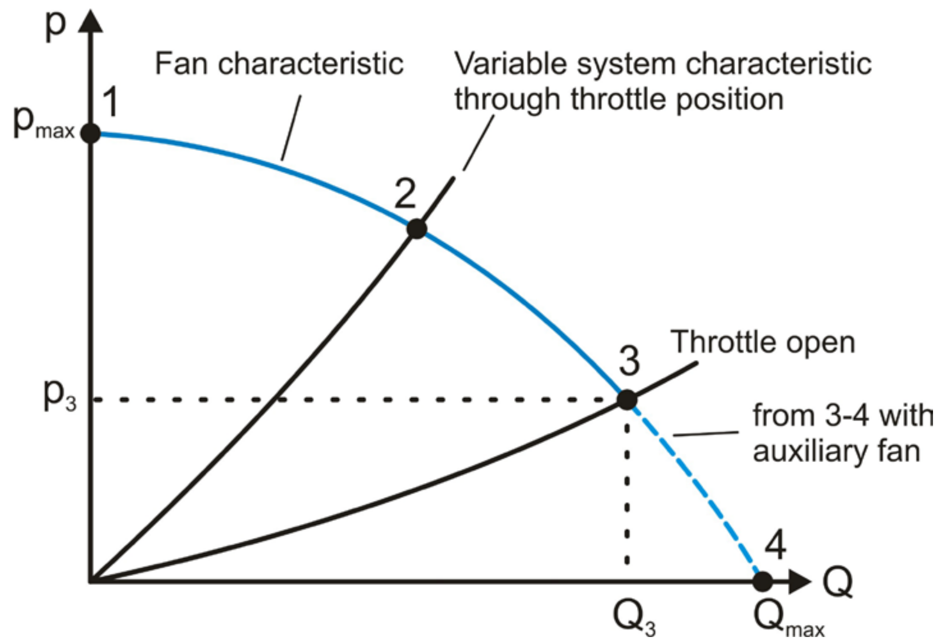
- Even with continuous inprovements of CFD (Computational Fluid Dynamics) EFD (Experimental Fluid Dynamics) is still basically part of the design or at least validation process of fans and blowers.
- Therefore, test rigs for fans and blowers are needed.
- These test rigs in general are unique and built according to a corresponding standard.
- These standards, as for example the German DIN 24 163 or the European DIN EN ISO 5801 prescribe only the main proportions of such test rigs while several important features are not described in detail.
- In particular the aerodynamic theory behind the standard is very often omitted.
- For example, the measurement of the pressure characteristic of a fan is performed at a pressure tab at the test chamber wall and not at the fan itself. How to assure that the pressure measured at the wall tab corresponds to the fan pressure?
- In this work the theory behind the design of test rigs was worked out for the relevant test rig features where the standards do not explain the fluid mechanical aspects.
- On this basis pressure and suction side test rigs were designed and completely simulated with a commercial CFD program, Ansys CFX – and built.



Fans and Blowers Characteristics

- The main performance characteristic of a fan is given by the total-to-static pressure raise Δp_{t-s} against the volumetric flow rate Q at a constant rotating speed n.
- The operating condition is varied with a throttle valve.
- The operating point of a fan is given by these three values, i.e. pressure rise, flow rate and rotating speed.

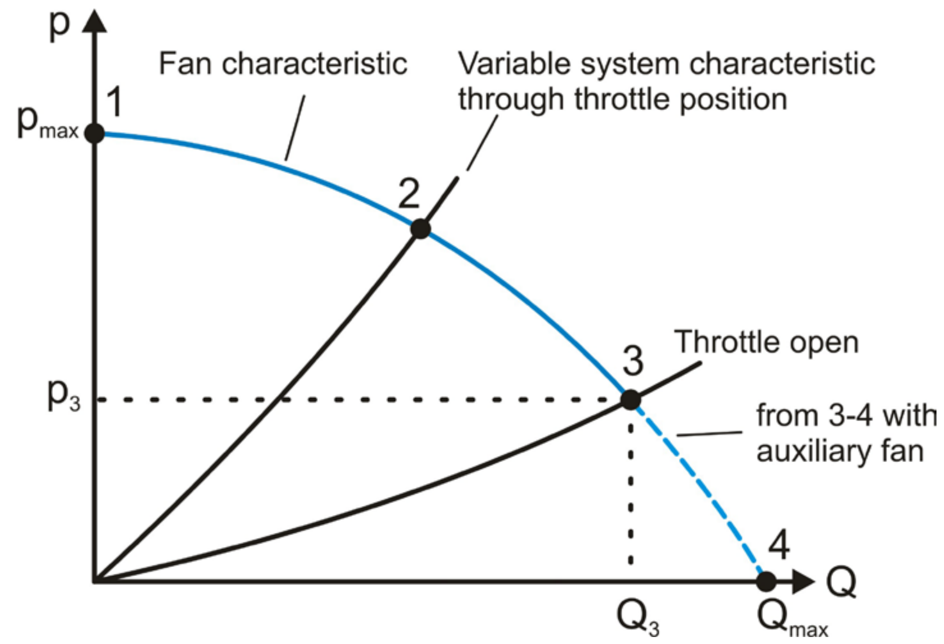


$$P_{hyd} = Q \cdot \Delta p_{t-s}$$

$$\Delta p_{t-s,\,fan} = \Delta p_{System} = k_{Rig} \cdot Q^2$$

Measurements

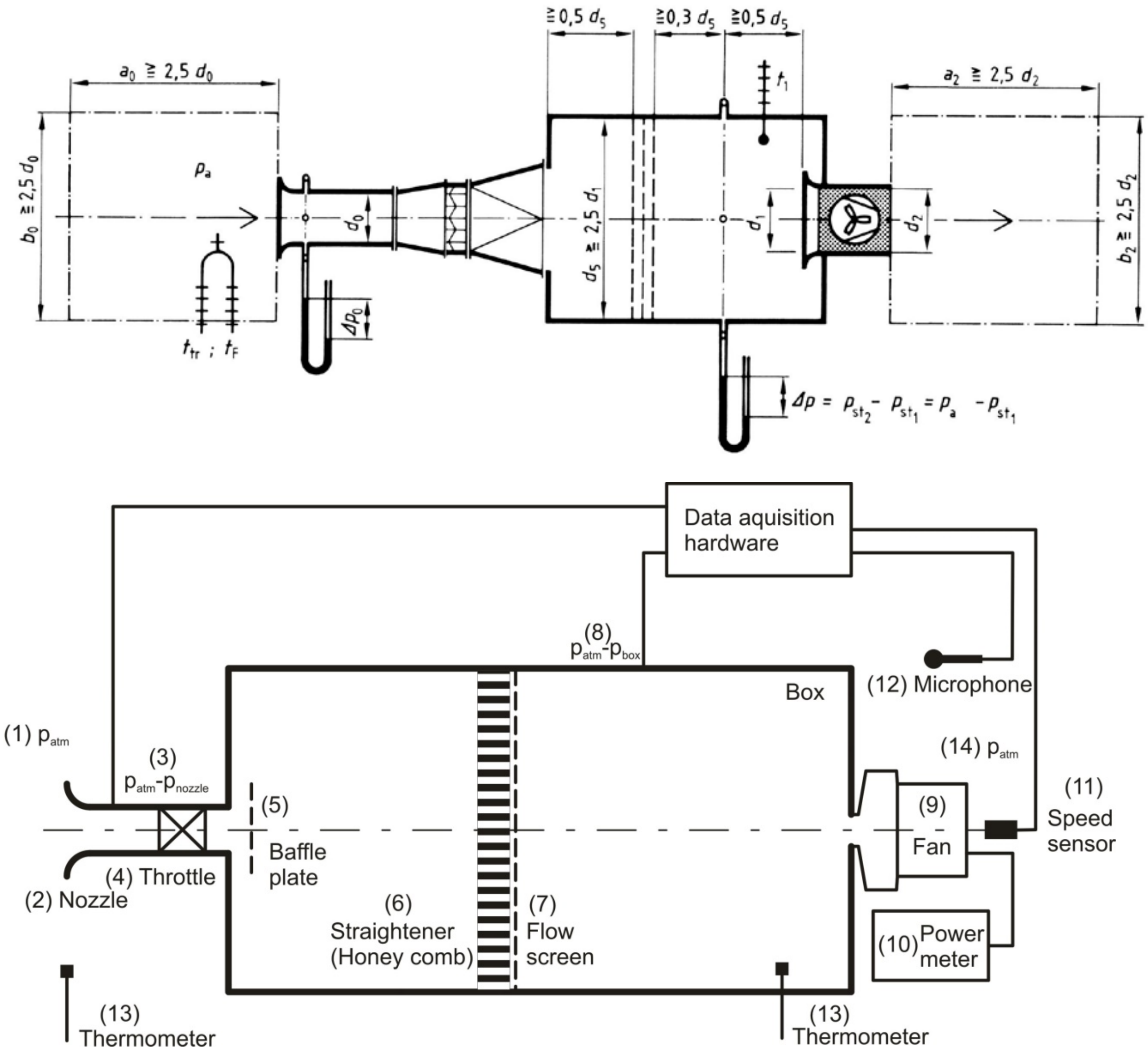
- Measurement of the fan pressure and flow rate are not done at the fan
- It is only possible to measure pressure and flow rate before or after the fan.
- Therefore a suitable system has to be used, in order to measure the proper values of pressure and flow rate.
- For this purpose test rigs are built.



$$P_{hyd} = Q \cdot \Delta p_{t-s}$$

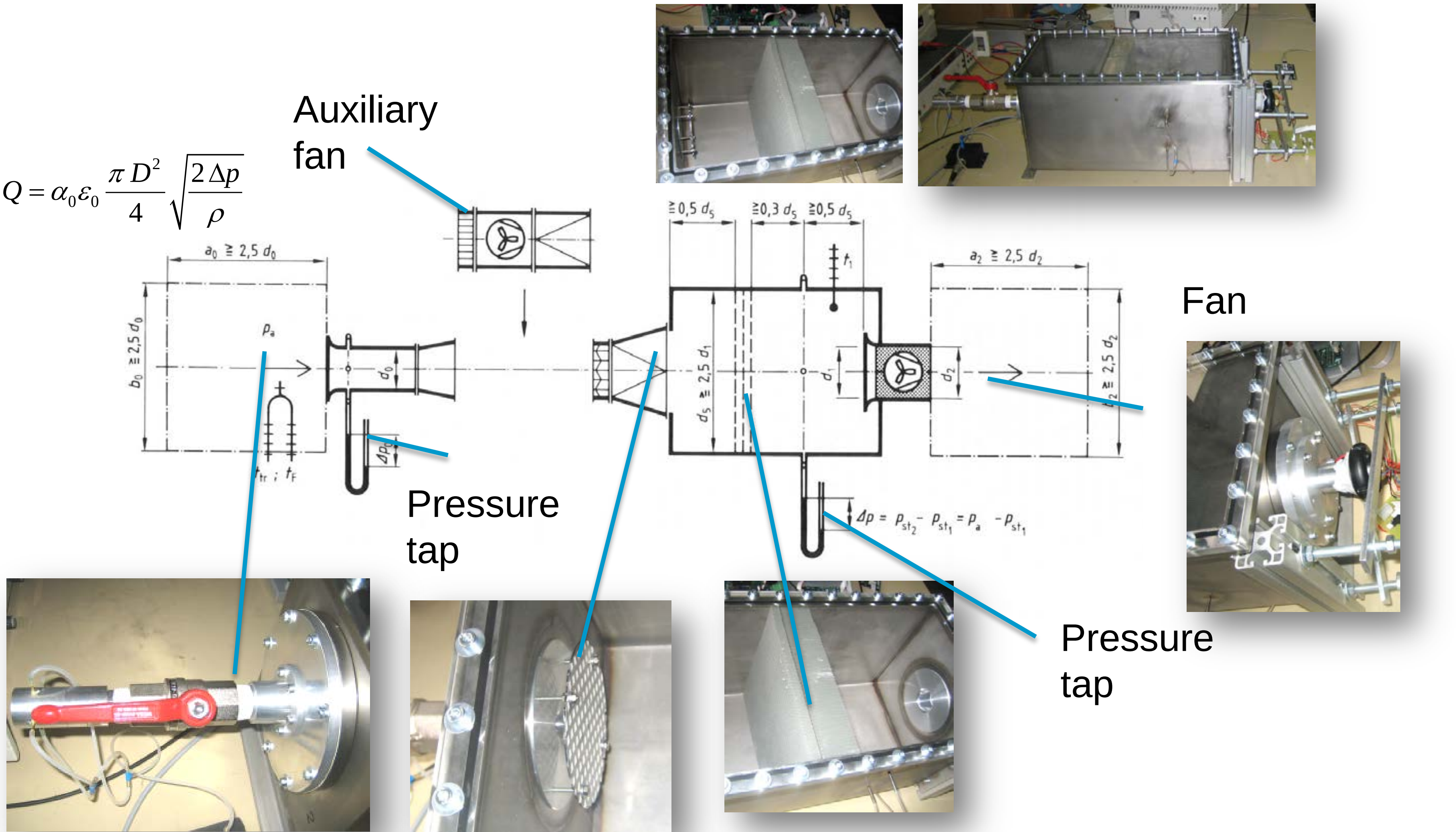
$$\Delta p_{t-s,\,fan} = \Delta p_{System} = k_{Rig} \cdot Q^2$$

Test Rig Overview I



Test Rig Overview II

$$Q = \alpha_0 \varepsilon_0 \frac{\pi D^2}{4} \sqrt{\frac{2 \Delta p}{\rho}}$$



Flow rate nozzle and throttle

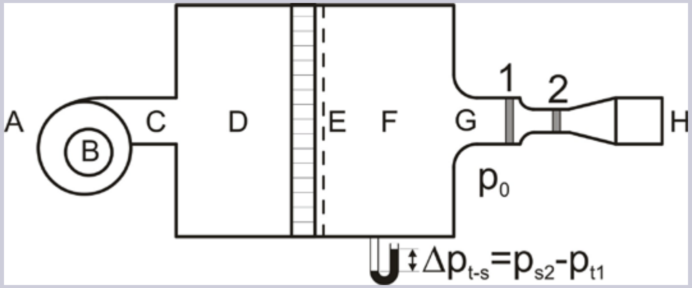
Baffle plate

Straightener and screen

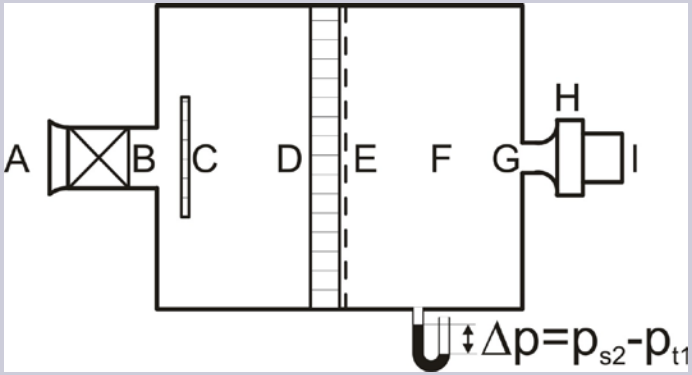
Pressure side and suction side test rigs

Fan and Blower Test Rig

Pressure side

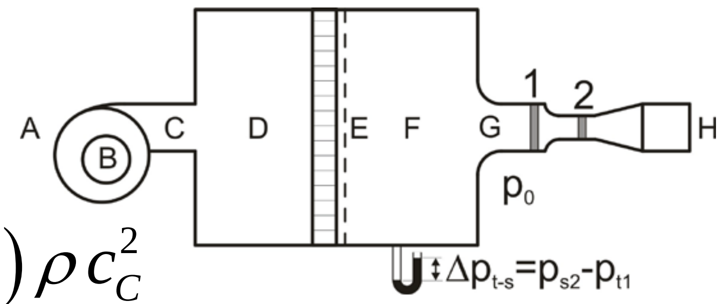


Suction side



Pressure side test rig

The key to the measurement of the total-to-static pressure of the fan in a pressure side test rig lies between C and D:



$$p_{tC} = p_{tB} + \Delta p_{t,\,fan} = p_{tB} + \Delta p_{t-s,\,fan} + \left(1/2\right)\rho\, c_C^2$$

Pressure increase due to area increase

$$\left(p_{sD} - p_{sC}\right)_{Area\ increase}$$

$$= \left(1/2\right)\rho c_C^2 \left(1 - c_D^2 / c_C^2\right)$$

$$= \left(1/2\right)\rho c_C^2 \left[1 - \left(A_C / A_D\right)^2\right]$$

Pressure loss due to sudden expansion

$$\left(p_{sD} - p_{sC}\right)_{Shock\quad losses}$$

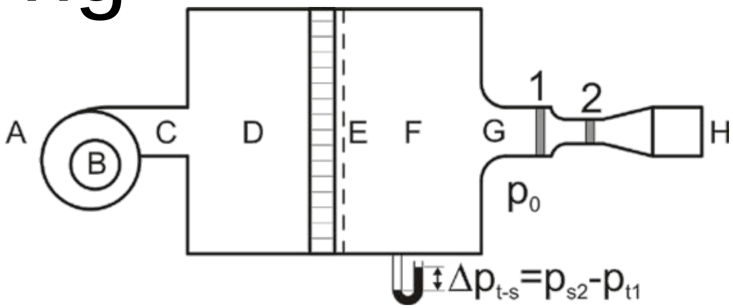
$$= \left(1/2\right)\rho\left(c_C - c_D\right)^2$$

$$= \left(1/2\right)\rho c_C^2 \left[1 - 2c_D / c_C + \left(c_D / c_C\right)^2\right]$$

$$= \left(1/2\right)\rho c_C^2 \left[1 - 2A_C / A_D + \left(A_C / A_D\right)^2\right]$$

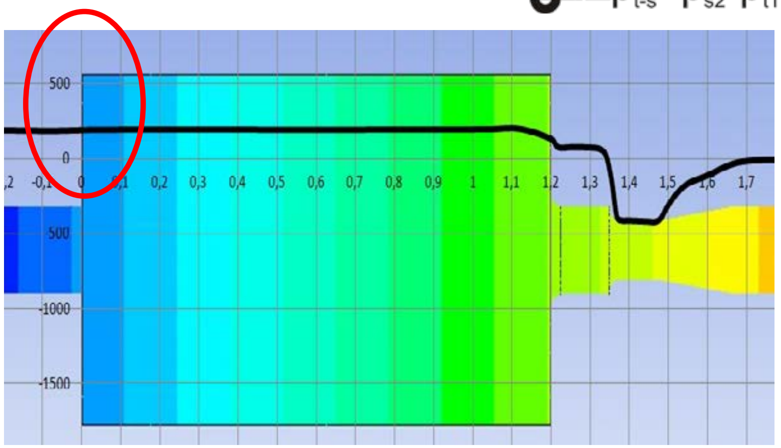
Pressure side test rig

The key to the measurement of the total-to-static pressure of the fan in a pressure side test rig lies between C and D:



Putting it together:

$$\left(p_{sD}-p_{sC}\right)_{Shock\quad losses}$$
$$=\left(1/2\right)\rho\left(c_C-c_D\right)^2$$
$$=\left(1/2\right)\rho c_C^2\left[1-2c_D/c_C+\left(c_D/c_C\right)^2\right]$$
$$=\left(1/2\right)\rho c_C^2\left[1-2A_C/A_D+\left(A_C/A_D\right)^2\right]$$

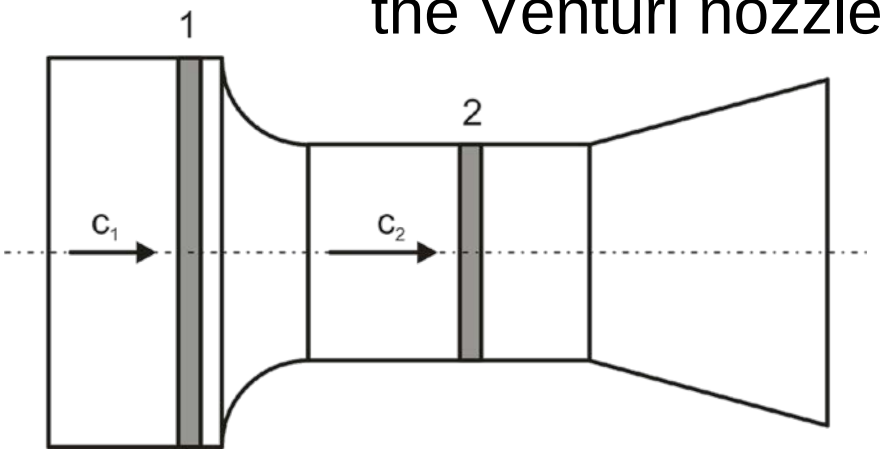


or $\left(p_{sD}-p_{sC}\right)_{Overall}=-\rho c_C^2\left[\left(A_C/A_D\right)+\left(A_C/A_D\right)^2\right]\approx 0$ for $A_C/A_D \ll 1$

Hence $\boxed{p_{sD}=p_{sC}}$ and therefore:

$$\boxed{\Delta p_{t-s,\,fan}=p_{sC}-p_{tA}=p_{sC}-p_0=p_{sF}-p_0}$$

Flow rate measurement: the Venturi nozzle



$$Q = \alpha A_2 U_2 = \alpha \frac{\pi d_2^2}{4} \sqrt{\frac{2(p_1 - p_2)}{\rho(1 - m^4)}}$$

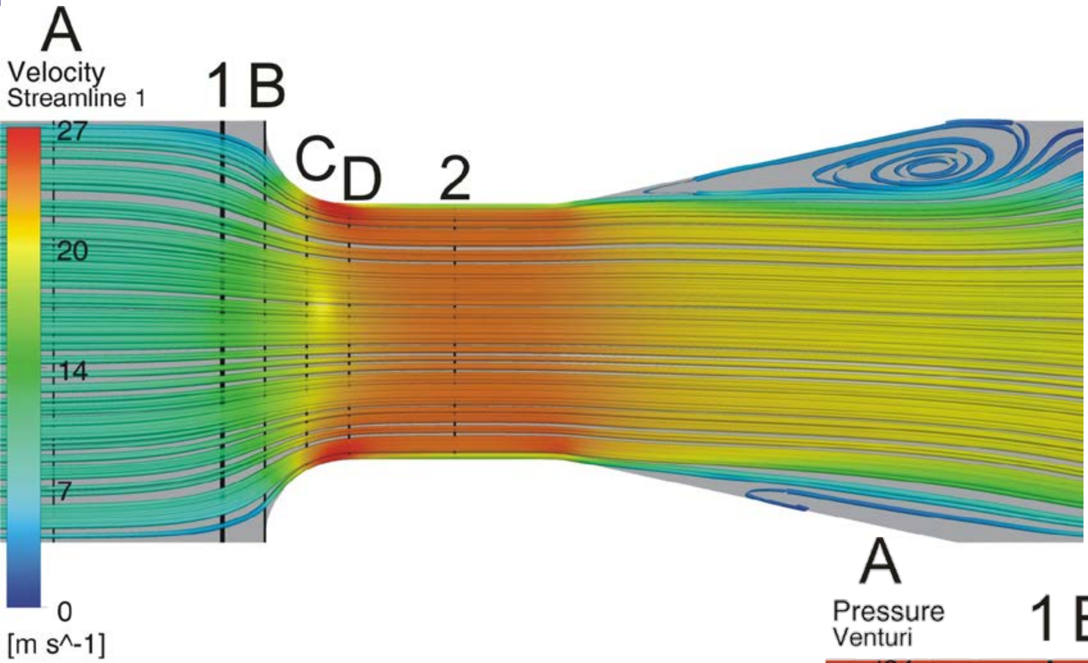
$$m = d_2 / d_1$$

The expansion factor α takes care of the fact that the flow is not one dimensional and not ideal:

- due to the development of the boundary layer at the walls, there is a displacement effect and the effective flow cross section is smaller than the geometrical cross section area, i.e. there is a blockage
- due to the constriction after position 1 the streamlines are extremely curved

Coburg University of applied sciences and arts

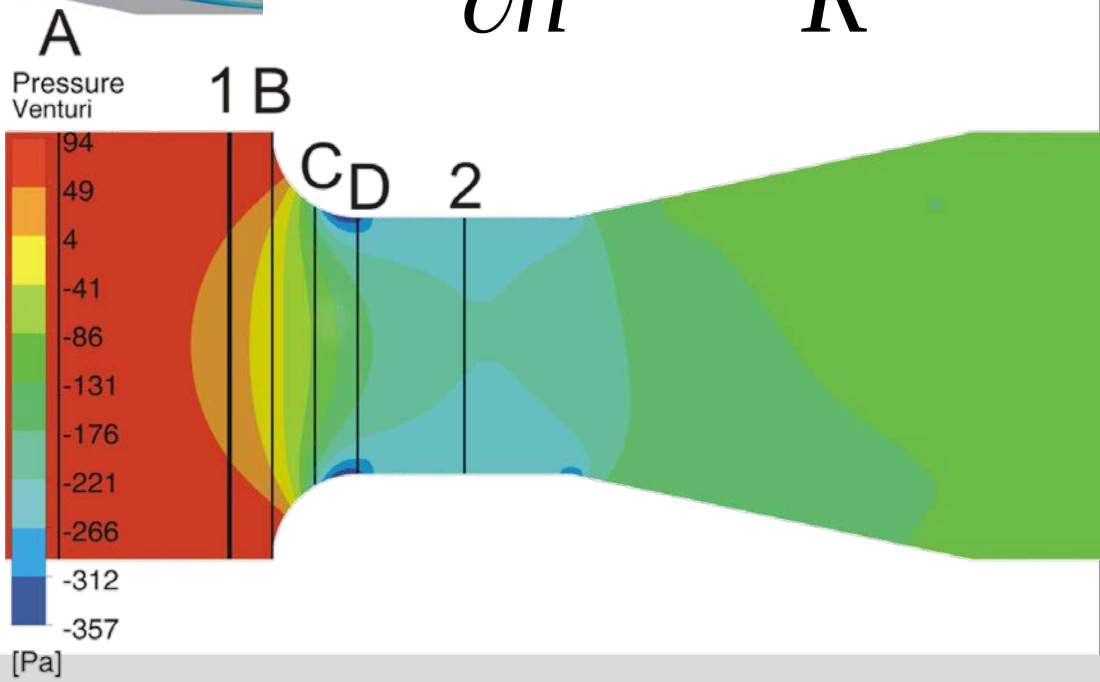
Effect of streamline curvature



Curved streamlines in the Venturi nozzle:

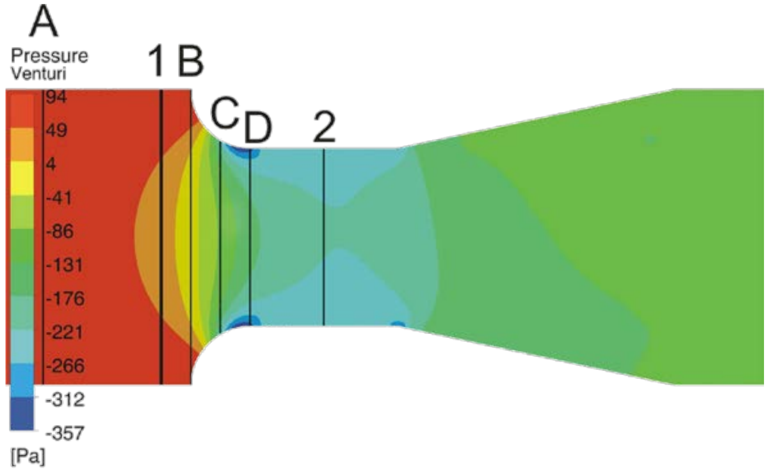
$$\frac{\partial p}{\partial n} = \rho \frac{c^2}{R}$$

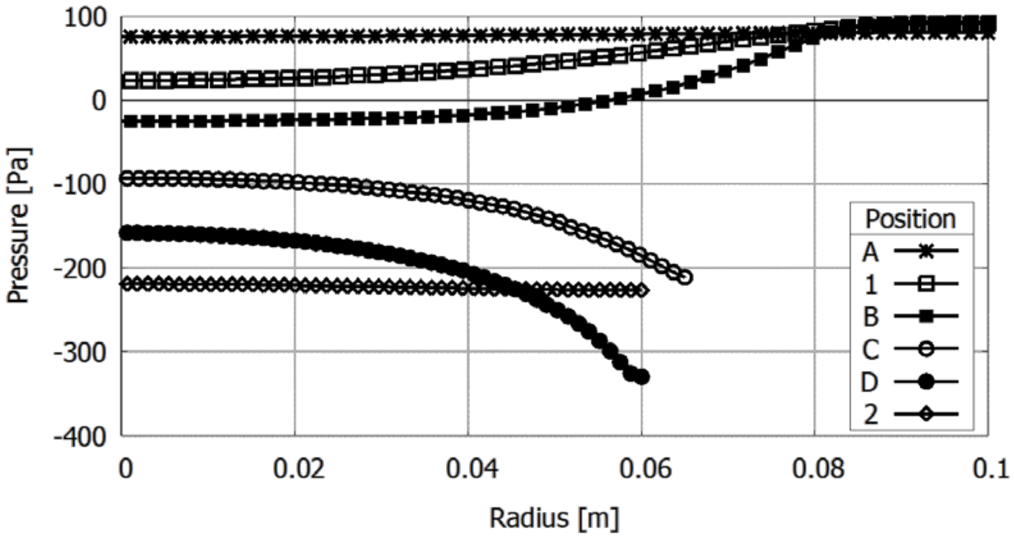
1. Curvature of the streamlines
2. Cross section area change

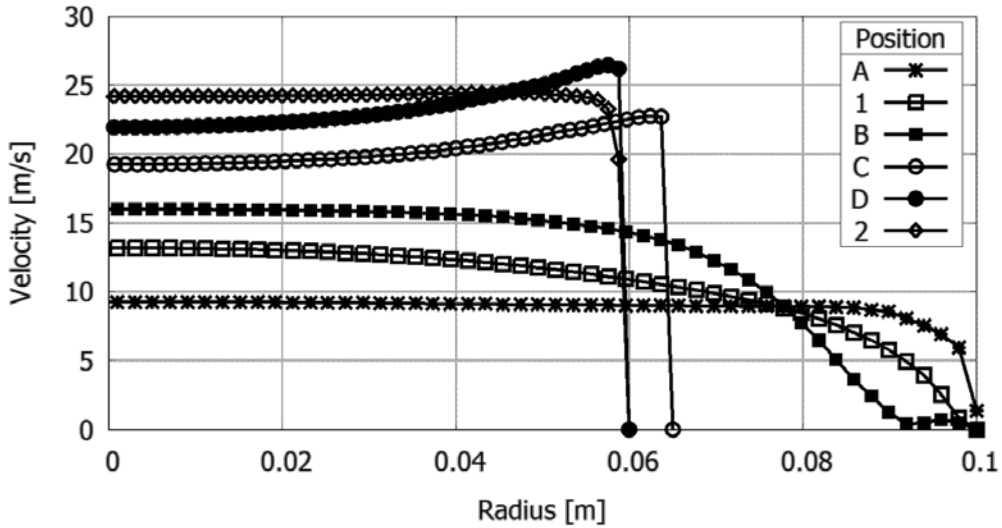


The pressure is not constant at a given cross section of the Venturi nozzle

Effect of streamline curvature



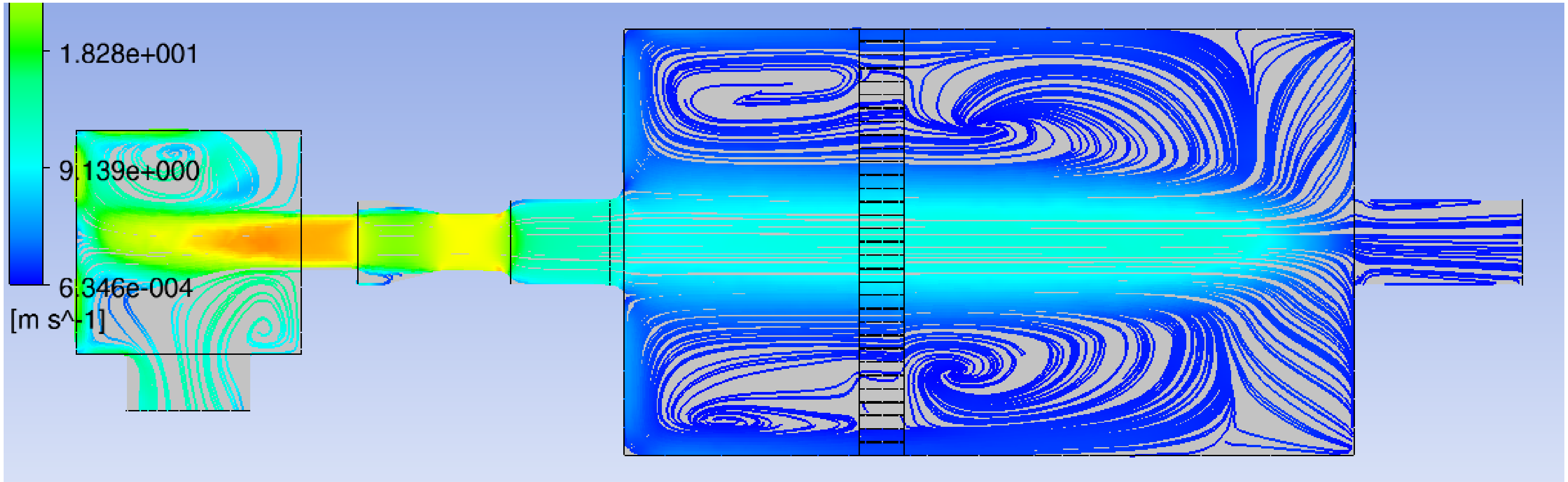




The velocity profile is approximately straight and the pressure constant in sections A and 2 but in between these sections the velocity profile is quite curved. Especially in section 1, where the pressure is also measured for the determination of the flow rate, one can see that it is not constant over the cross section.

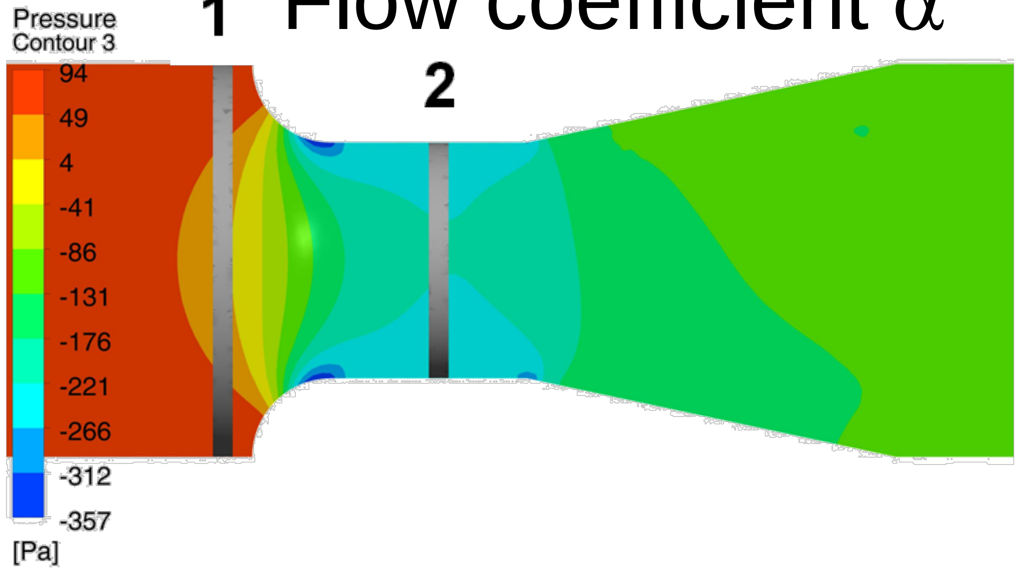
Flow coefficient α

It is important to determine the flow coefficient α in order to compare measurements of different Venturi nozzles or even test rigs. The flow coefficient α can be determined with CFD simulations.



Flow coefficient α

$\dot{m}_{BC}$



1. inlet boundary condition set to mass flow rate
2. Computation of the flow coefficient α as

$$\boxed{\alpha = \frac{\dot{m}_{BC}}{\rho \frac{\pi d_2^2}{4} \sqrt{\frac{2\left(p_1 - p_2\right)}{\rho\left(1 - m^4\right)}}}}$$

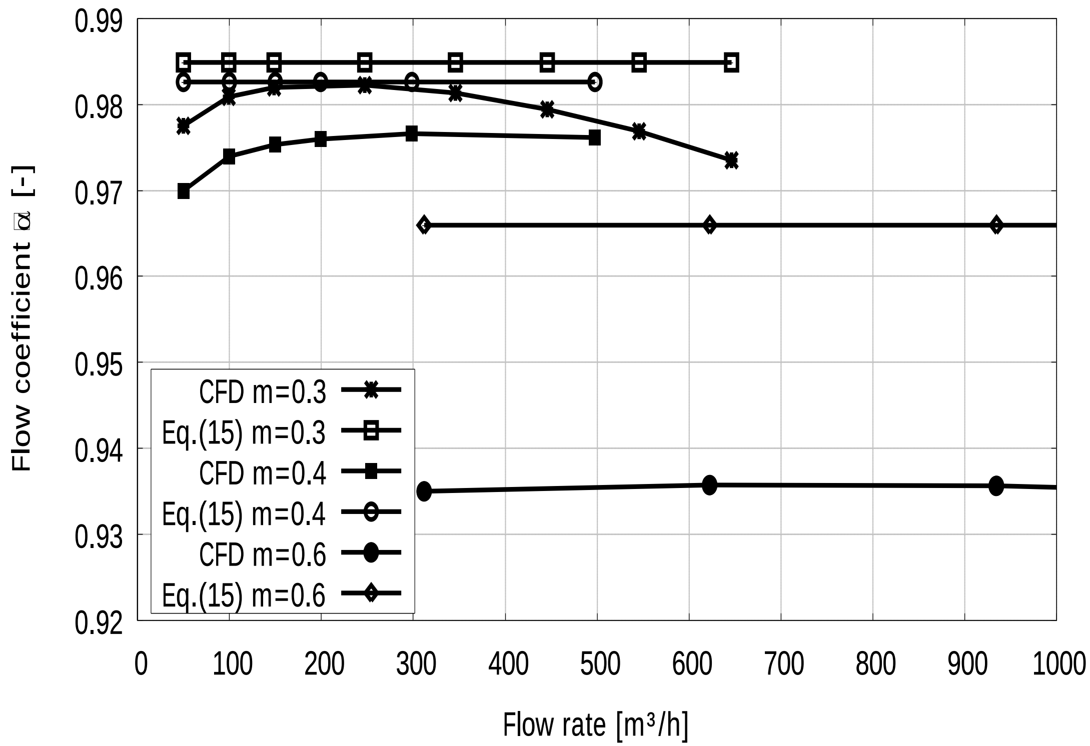
The pressures have to be evaluated at the surface of the Venturi nozzle rather then across the section in position 1, since on the measurements the pressure is also taken at the wall only.

Flow coefficient α

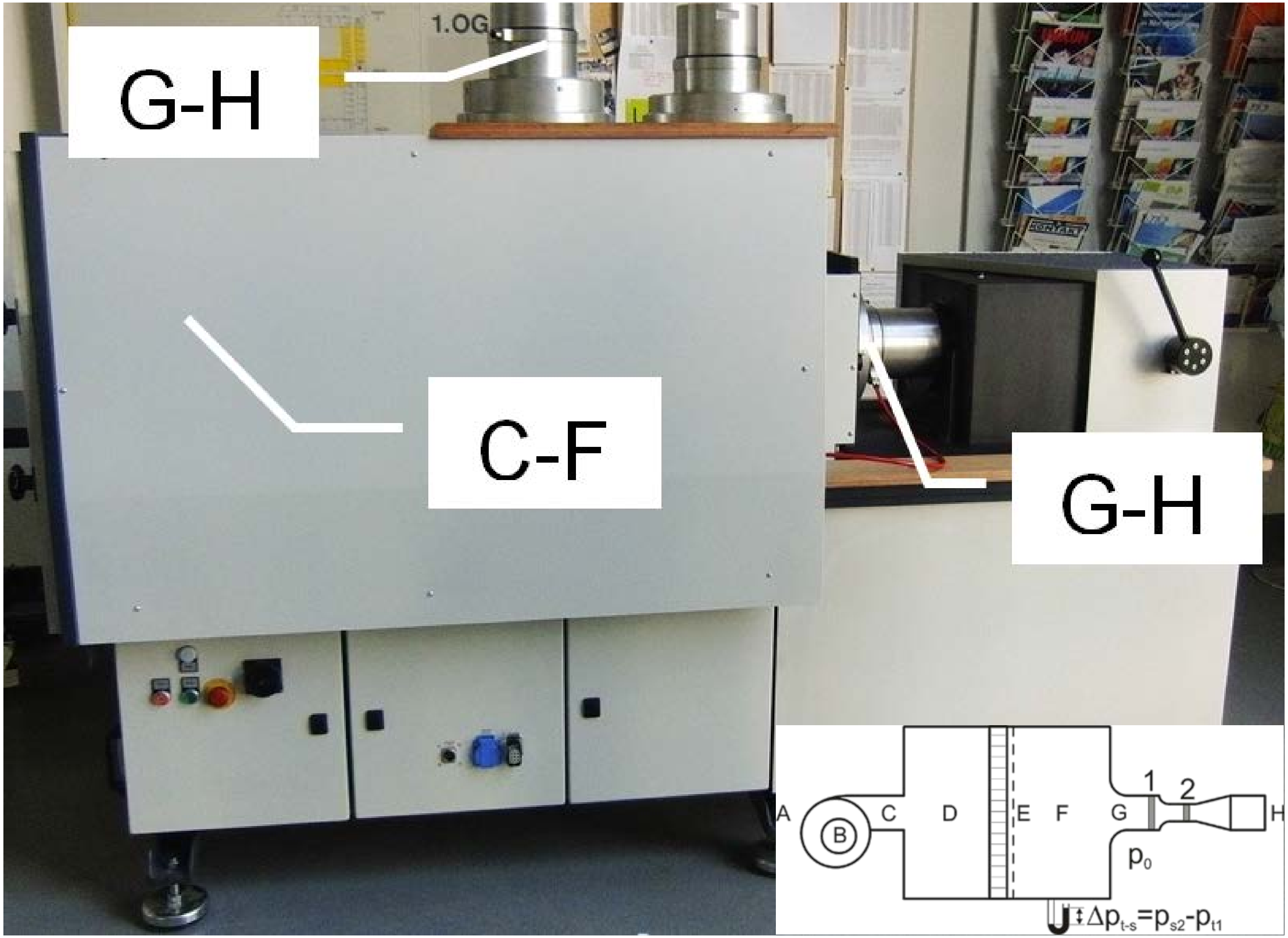
- There is a dependency on the flow rate or on the Reynolds number which cannot be predicted by equation
- The tendency, however, that the flow coefficient is lower the higher the diameter ratio m can be seen in both cases.
- flow coefficients as predicted by CFD ensure that the flow rate measured by Venturi nozzles with different diameter ratios m is the same.

DIN EN ISO 5167-3 (empirical formula):

$$\alpha = 0.9858 - 0.196 \cdot m^{4.5} \quad m = d_2 / d_1$$



Pressure side test rig

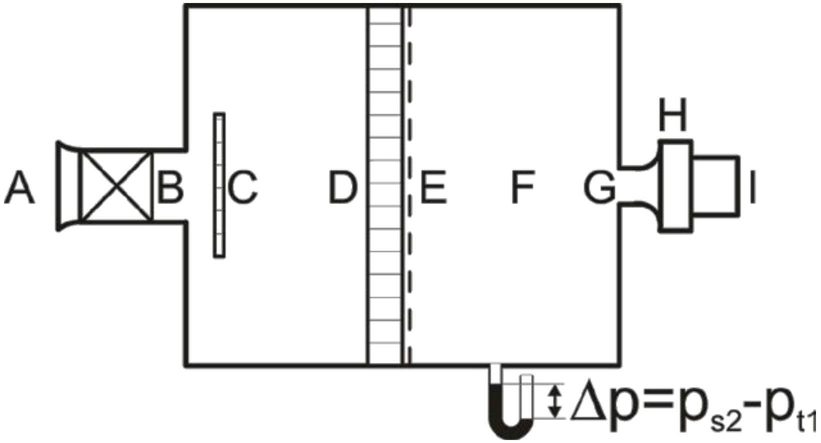


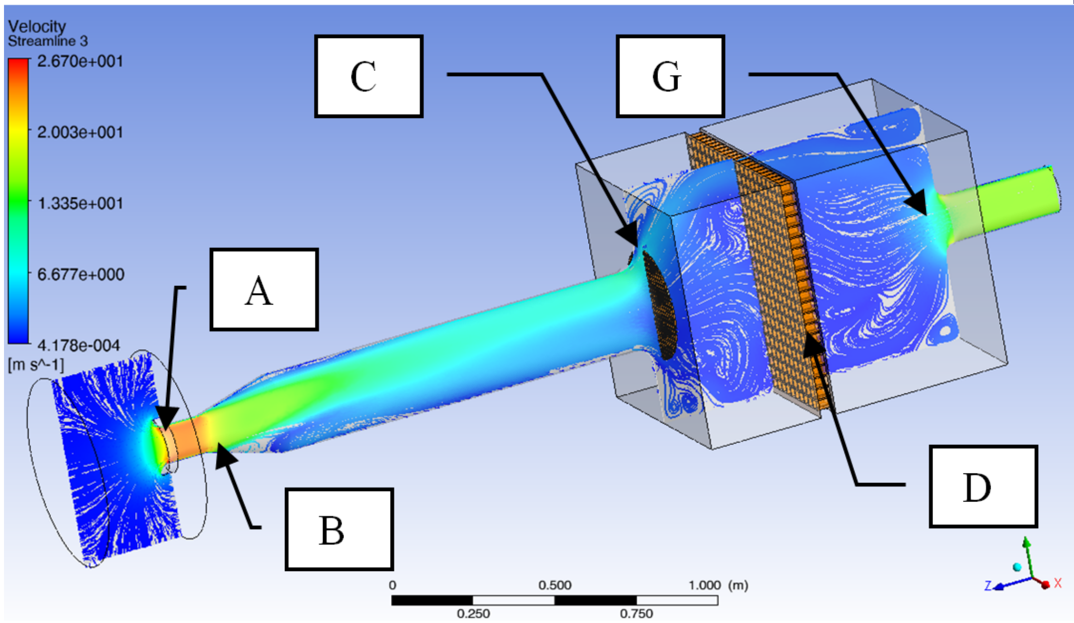
Suction side test rig

Working principle:

$$p_{tF} = p_{sF} + (1/2)\rho c_F^2 \approx p_{sF}$$

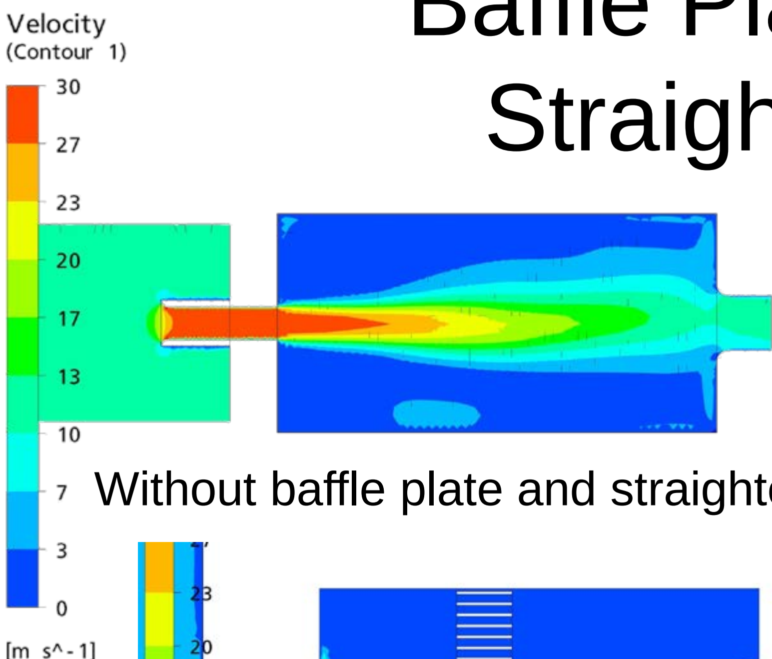
$$\Delta p_{t-s,\,fan} = p_{sF} - p_0 = p_{s2} - p_{t1}$$



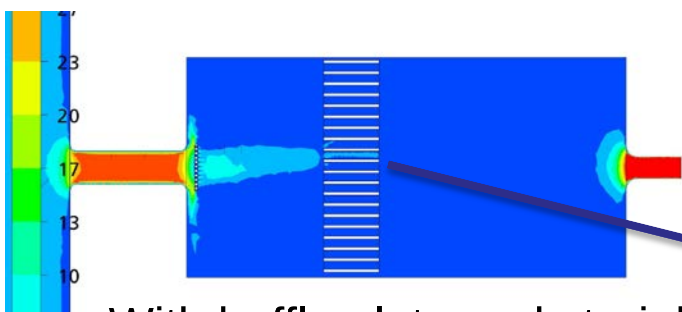


COBURG UNIVERSITY
of applied sciences and arts

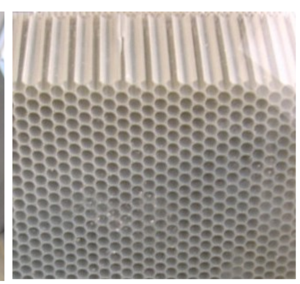
Baffle Plate and Straightener

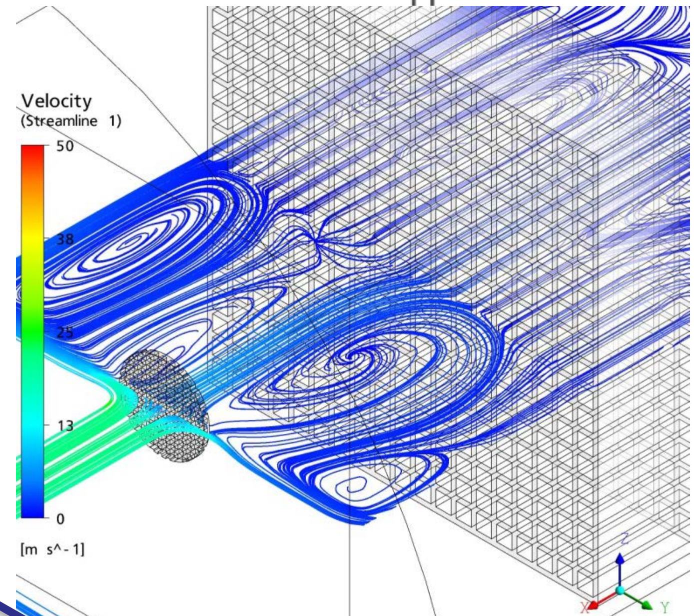


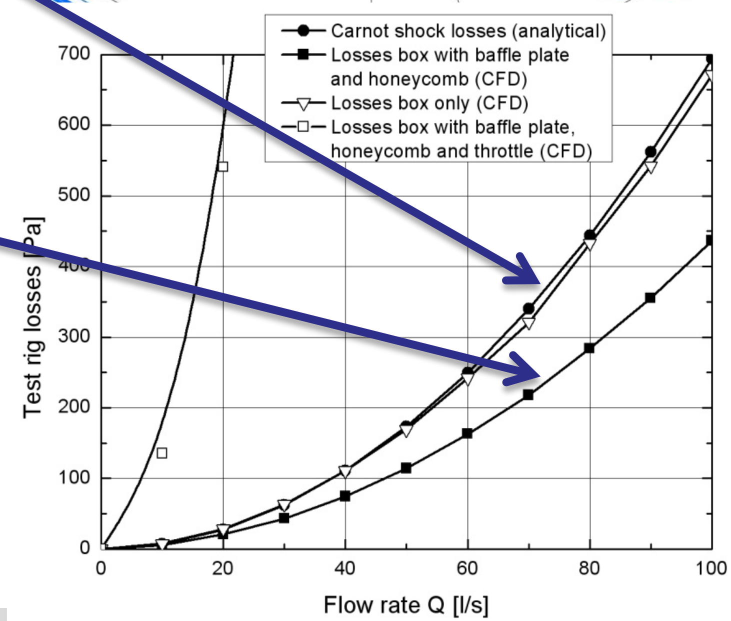
Without baffle plate and straightener



With baffle plate and straightener







Flow coefficient α

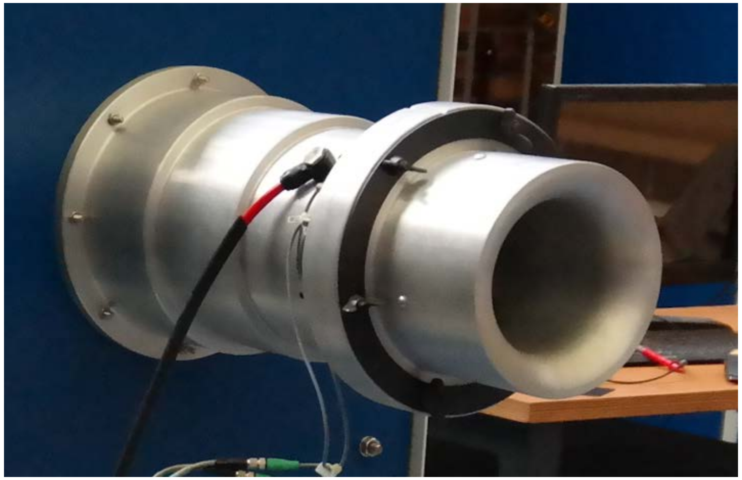
$$Q = \alpha A_{nozzle} \sqrt{2(p_0 - p_{nozzle})/\rho}$$

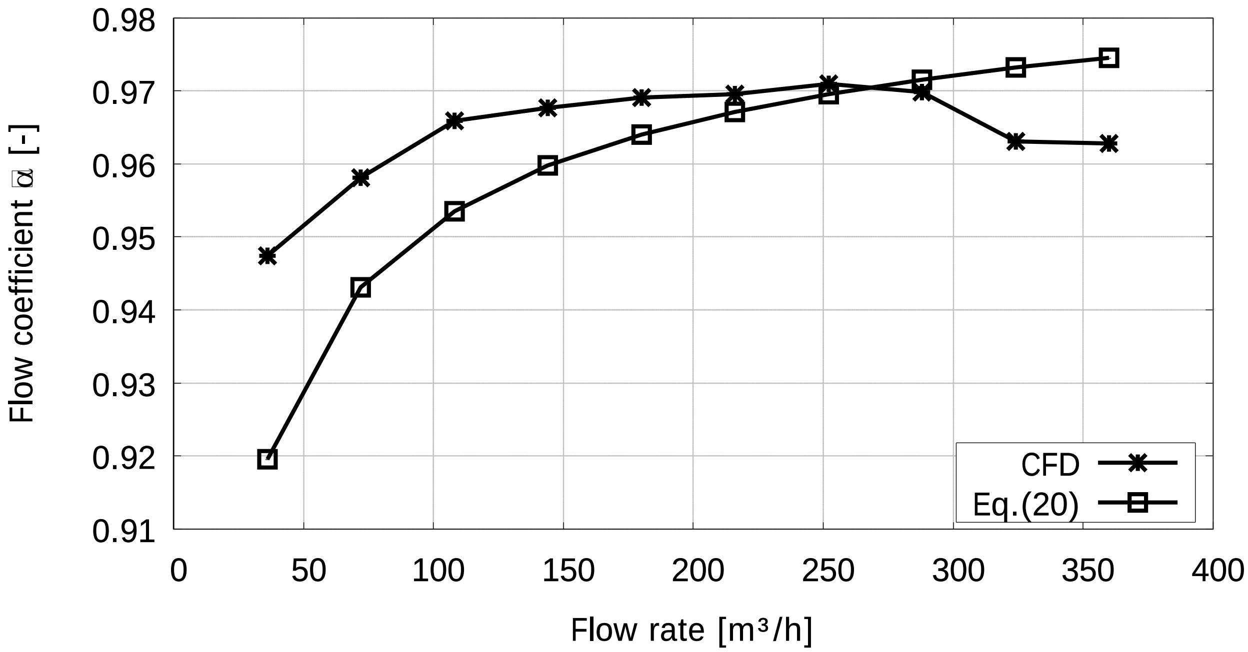
German Engineering Standard
DIN 24 163 part 2:

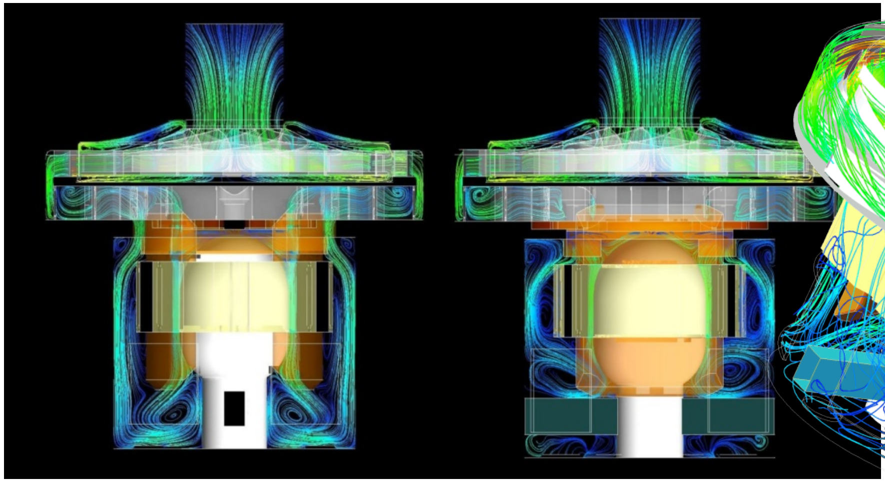
$$\alpha = 1.000 - 0.004\sqrt{10^6/\mathrm{Re}}$$

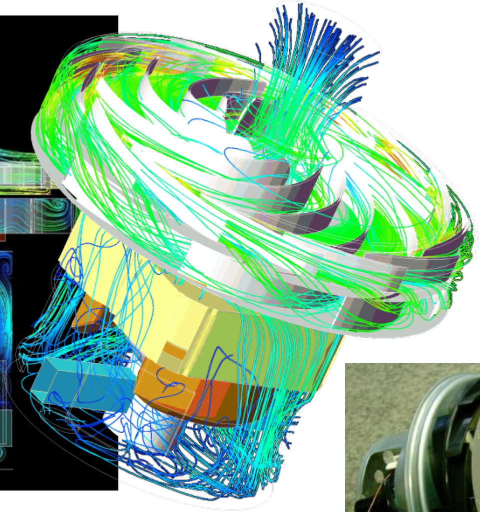
$$\mathrm{Re} = c\,d/\nu$$

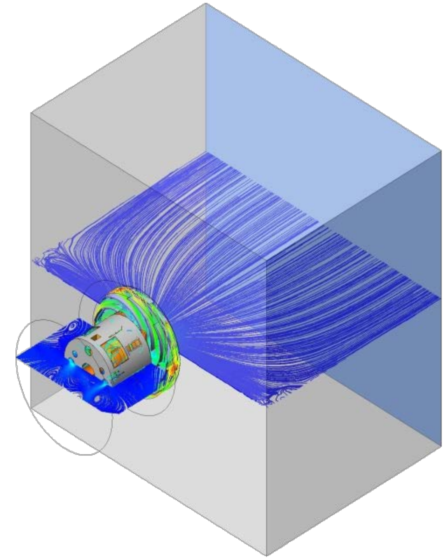
Quite good agreement of the empirical equatuion with the CFD results.

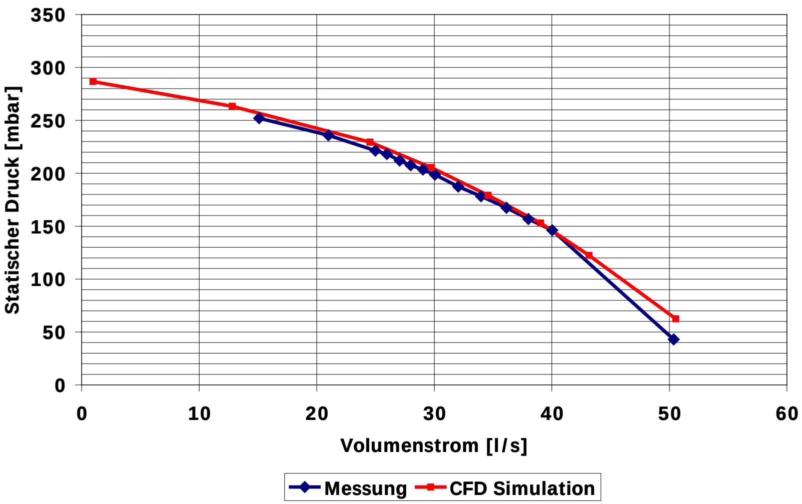


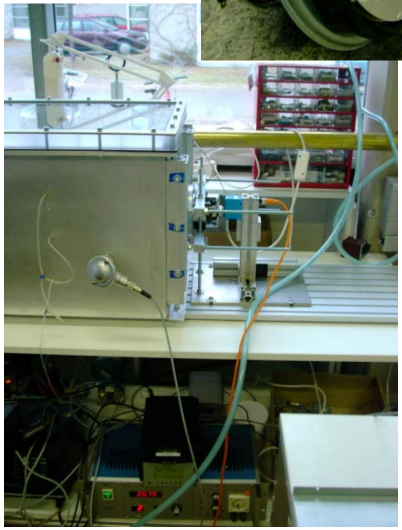


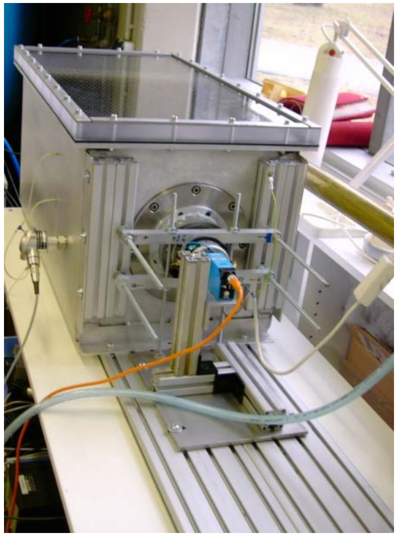








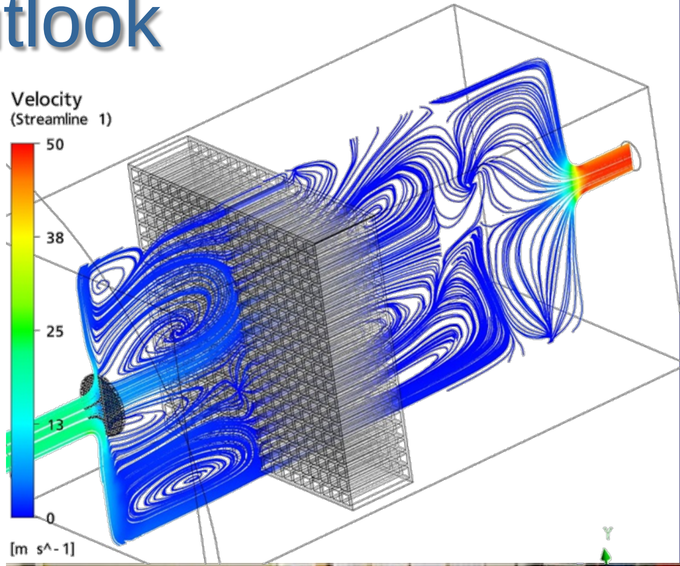


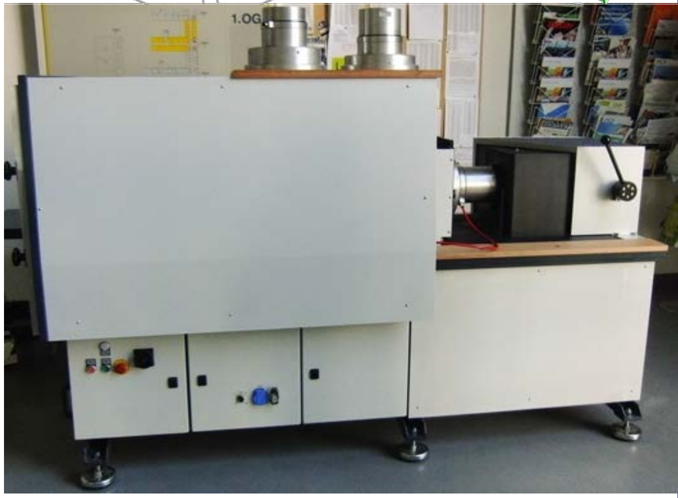


CFD and Measurements

Conclusions and Outlook

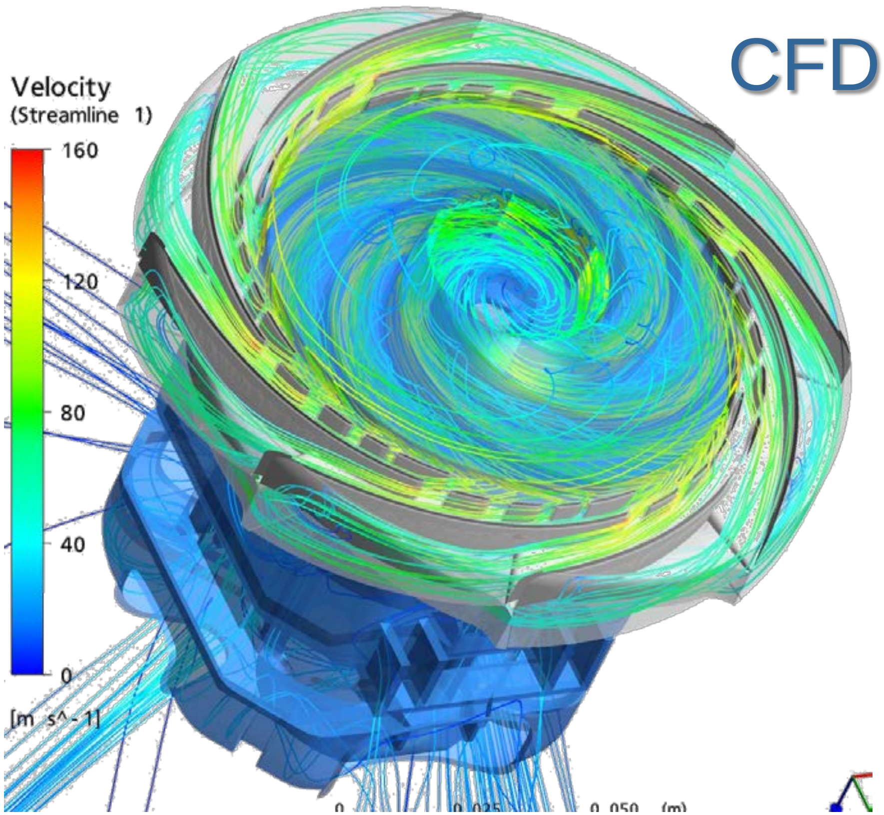
- In this work the instructions from the standards, analytical computations as well as CFD computations were combined in order to reach an optimum and compact test rig design
- flow coefficients, which usually are determined by empirical correlations, were precisely computed with CFD
- for Venturi nozzles the empirical correlations for the flow coefficient of the standards are not allways accurate.
- In particular they do not include the effect of the Reynolds number.
- In that case the flow coefficients as computed by CFD deliver much more reliable results.
- In order to validate the test rig designs two test rigs were built, one suction side and the other pressure side.





Thank you!
Any questions?

CFD



EFD

REFERENCES

1) **<u>Case study I</u>**:
Epple, Ph., Miclea, M., Ilic, C., Delgado, A.: *Combined Impeller-Diffuser Design and the Influence of Slotted Guide Vanes on the Performance of Radial Diffusers*, ASME International Mechanical Engineering Congress and Exposition, Orlando, 2009.

2) **<u>Case study II</u>**:
Epple, Ph., Miclea, M., Pfannschmidt, K., Grobeis, D., Delgado, A.: *A Design Method of Radial Fans Considering the Torque-Speed-Characteristic of the Motor*, ASME International Mechanical Engineering Congress and Exposition, IMECE2010-39050, Vancouver, 2010.

3) **<u>Case study III</u>**:
Epple, Ph., Semel, M., Willinger, B., Delgado, A.: Compact Test Rig Design For Fans And Blowers, International Mechanical Engineering Congress & Exposition IMECE2014, Nov 14-20, 2014, Montreal, USA.